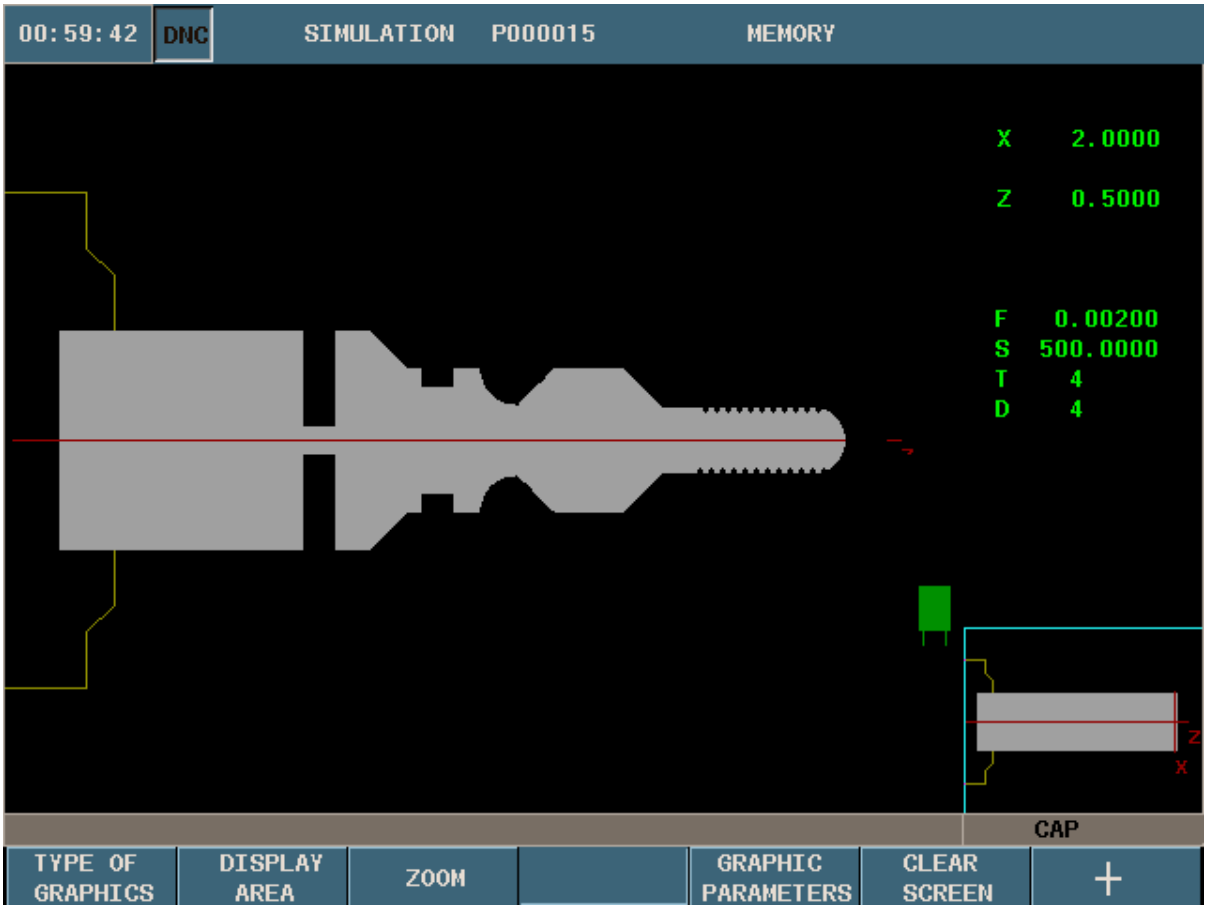
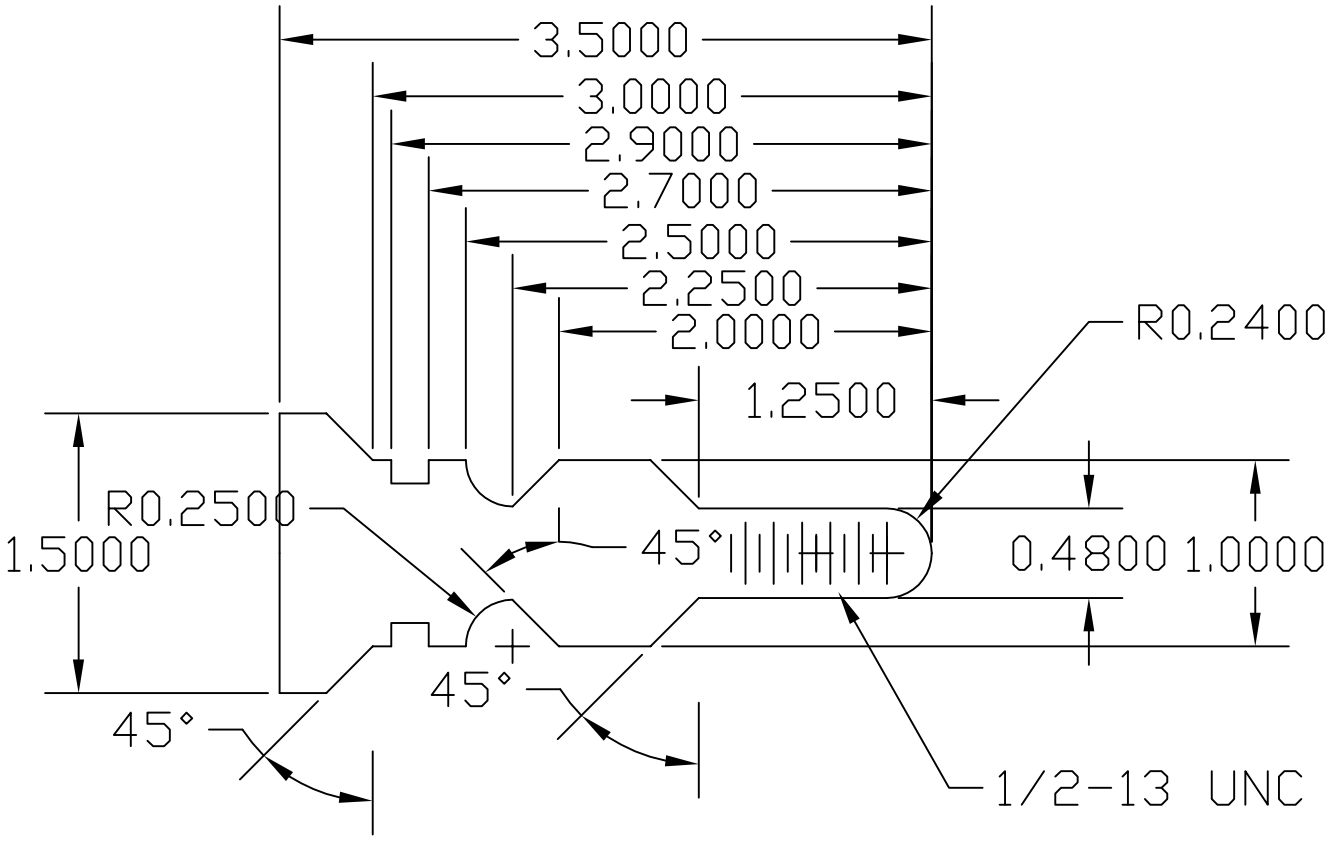
FAGOR AUTOMATION
TC TRAINING MANUAL



REVISIONS				
XXX13	XXX14	XXX15	XXX16	XXX17



	\			
	Fagor 8055 TC Demo Part			
	SIZE A	FSCM NO 0	DRAWING NUMBER 0001	REV 0
	SCALE 1		SHEET 1	

INDEX

Introduction……..……..3

Reference Search……..4

Tool Calibration……....5

Manual Preset…….……..7

Facing Cycle……..……..8

Simulation…….……...10

Storing Part Program……11

Turning Cycle 1……..…….13

Taper Cycle 1……..16

Turning Cycle 2……..…….19

Taper Cycle 2……...20

Rounding Cycle……...….25

Threading Cycle……..…….28

Profiling Cycle……..…31

Grooving Cycle…….35

Cut-Off Cycle……38

Part Drawing……..41

Zero Offsets……..43

ISO Mode…….45

Inserting Code In-Between Canned Cycles…….…….46

Tool Inspection…….48

Quick Reference Guide…….51

INTRODUCTION

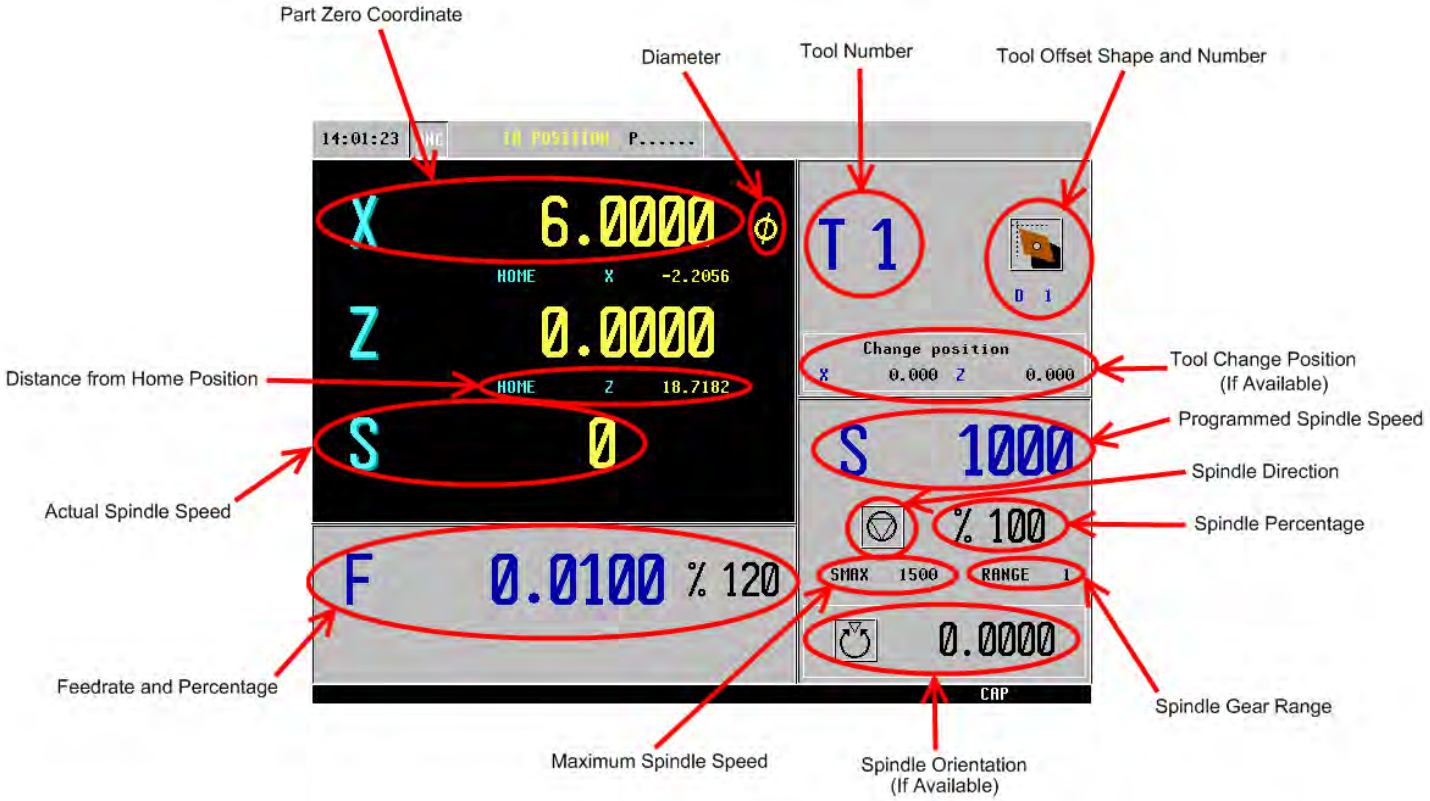
This manual has been generated to be a guideline for your assistance for the conversational programming of the Fagor control. In this manual we will be generating a part-program using the canned cycles that will be the most beneficial for your programming needs. In this manual we will be exercising all of the main canned cycles including:

- Tool Calibration
- Turning Cycle
- Facing Cycle
- Taper Cycle
- Rounding Cycle
- Threading Cycle
- Grooving Cycle
- Profiling Cycle

Along with some of the other operations that will be important to no how to use:

- Homing of the Axes
- Presetting your Zero
- Simulation/Graphics
- File Maintenance

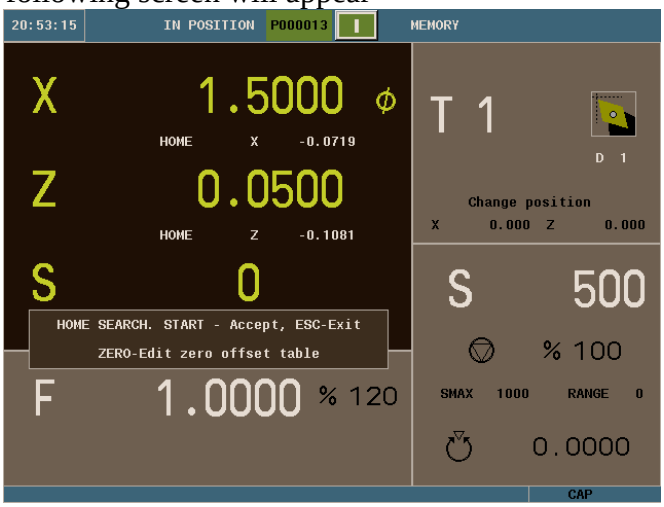
After the completion of these operations, you will begin to understand all the steps that are necessary in programming the Fagor CNC Control. Once you grasp each of these concepts you will be able generate any part in which you desire. Using this manual as your guideline will also help you generate part-programs in a timely manner. You will soon realize once you learn the programming of one canned cycle, you will understand the programming concept of each and every one of the other canned cycles. Whether it is your first time using a CNC or if you are a CNC programming expert, the conversational programming of the Fagor CNC Control will benefit all users.



REFERENCE SEARCH

On power-up, it is always recommended to home your machine before performing any specific task. This is more of a safety precaution. When referencing your machine the CNC will know exactly where it is located. To reference your machine perform the following operation:

1. From the Jog Screen select the Zero Icon
   - The following screen will appear



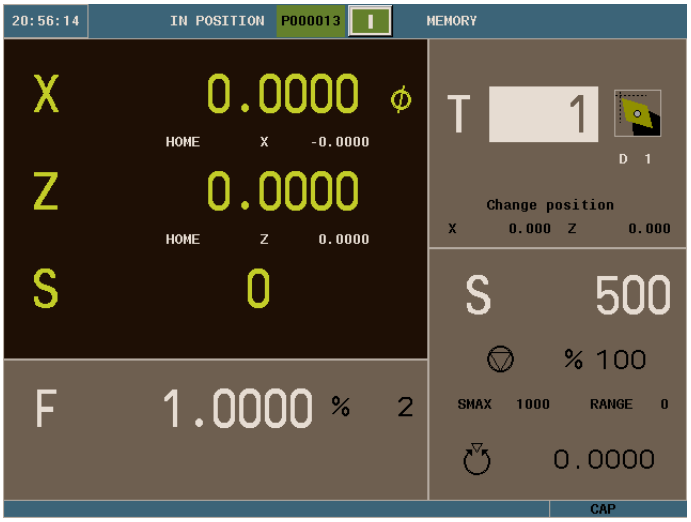
** Note: Pressing the ESC Key instead of Cycle Start will cancel the operation*

2. To execute the Home Search press cycle start
   - At this time the CNC will execute the subroutine associated with the home search.
   - Both axes will now move and find there home positions.

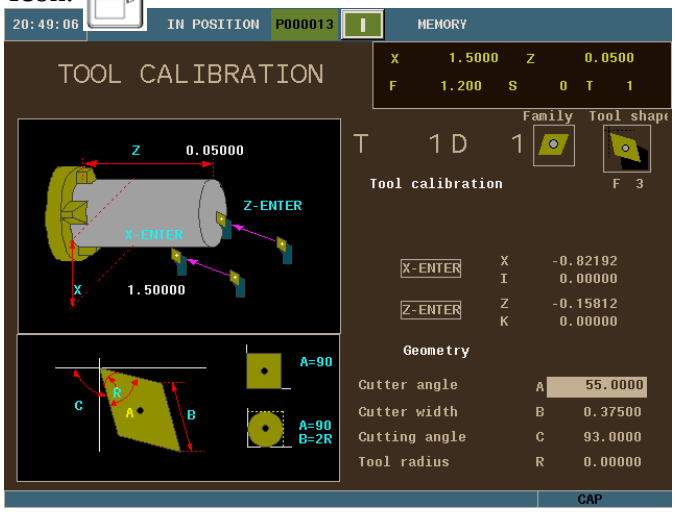
TOOL CALIBRATION

For this specific program we will be using four tools. Before we begin any type of programming we will calibrate all four of our tools on both axes.

1. From the Jog Screen enter the following commands:
   - T (the box will highlight in red)
   - 1 (Tool 1)
   - Cycle Start



2. The axes will now move to its Tool Change Position.
3. Select the F1 Icon.



4. The first section that is highlighted red is for the diameter of your part. In this example we will use 1.5 for X and .05 for Z.

5. Select your tool family by using the half-key to toggle through them. We will be using the diamond shape for tool 1.

6. Select the tool shape by using the half-key. We will use for tool 1.
7. Jog your X axis accordingly until the tip of your tool is touching your part.
8. Tool calibration:

   - Press X. The X-ENTER symbol will highlight red. Then press ENTER. This will automatically put your tool offset in X.
   - I- This is the tool offset wear for X axis. At this time it is unnecessary to put any value in this field.

9. Jog your Z axis accordingly until the tip of your tool is touching your part.

   - Press Z. The Z-ENTER symbol will highlight red. Then press ENTER. This will automatically put your tool offset in Z.
   - K- This is the tool offset wear for Z axis. At this time it is unnecessary to put any value in this field.

10. Geometry:
    - Enter the appropriate information for defining your tool geometry.
    - For tool 1 we will use the following example:

A: 55
B: 0.375
C: 93
R: 0.000

<u>TOOLS 2 , 3 , 4</u>

- Select T (Tool)
- Enter 2 , 3 , 4 (Tool Number)
- Press Cycle Start
- Repeat steps 5-10.

Note: It is very important that you define your tool geometry correctly.

Figure 1
(Profiling Tool)

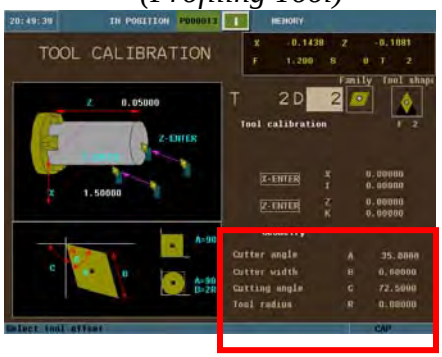


Figure 2
(Threading Tool)

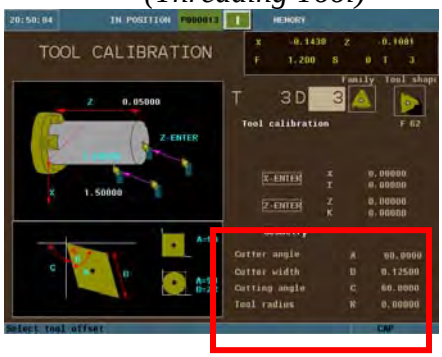
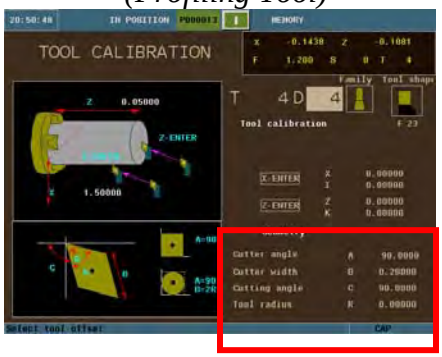


Figure 3
(Profiling Tool)



MANUAL PRESET

Note: Not required if using the tool calibration.

1. From the Jog Mode, jog your X axis until you are at the desired position. Enter the following keys:
    - X (select your axis)
    - The number (your diameter preset value) 1.5”.
    - Enter (confirming your position)
    - Enter (executing your command)

        - *Repeat these steps for your Z axis*

Figure 1

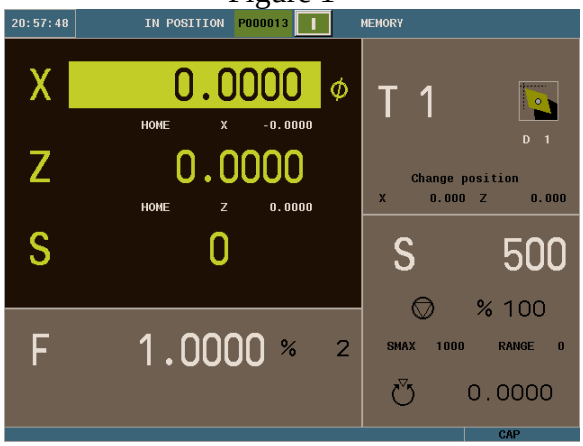


Figure 2

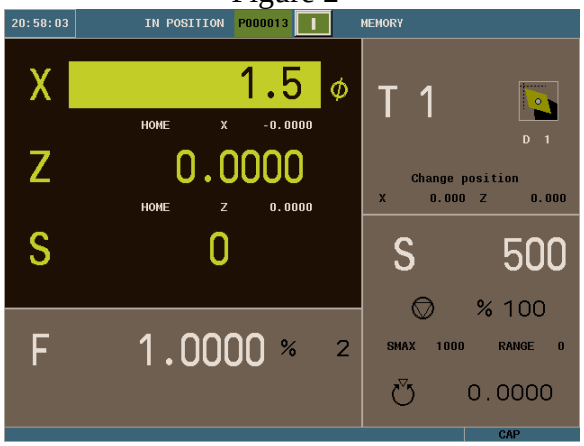


Figure 3

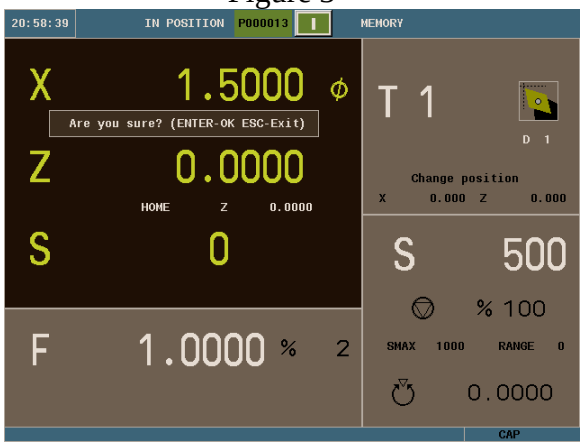
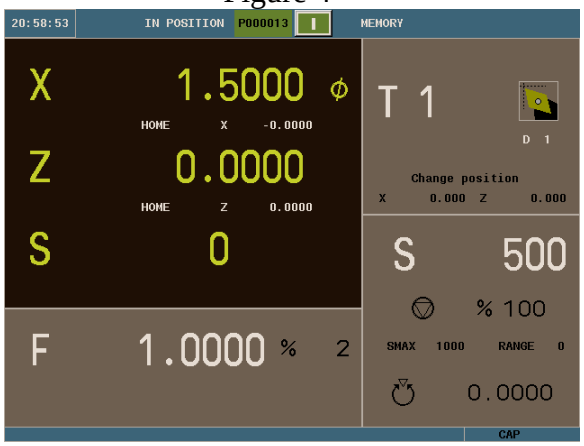
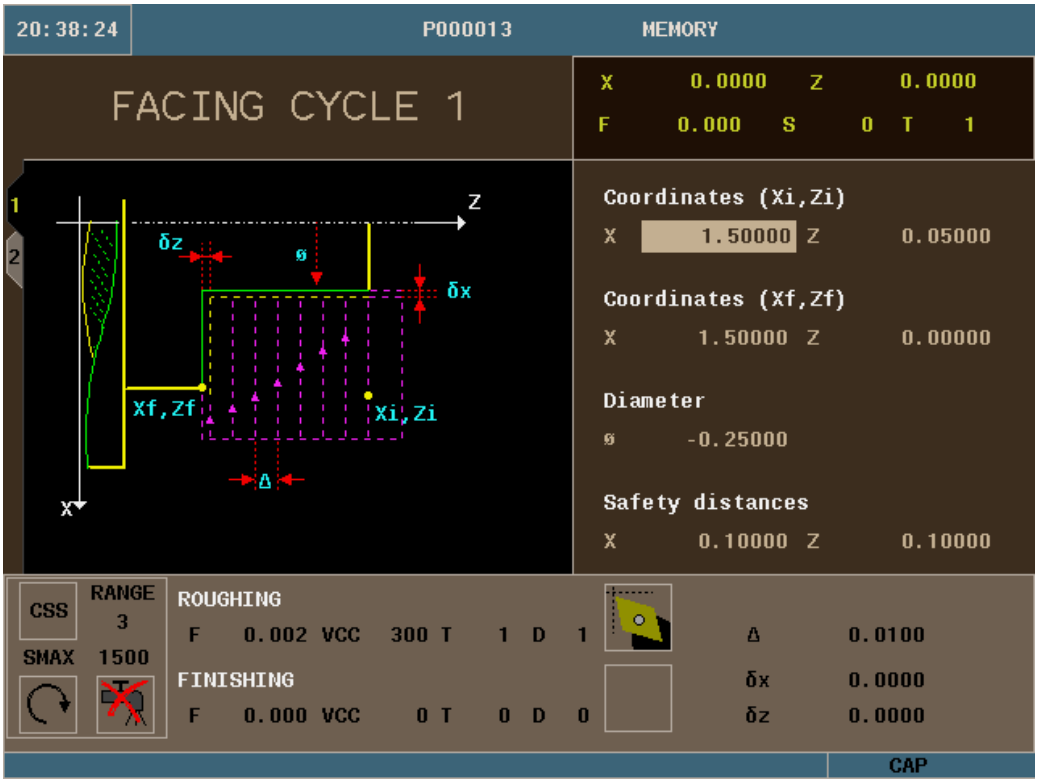


Figure 4



FACING CYCLE



1. Select the F3 Icon.
2. Define your part:
   - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1.5 and Z0.05
   - (Xf, Zf)- This is your ending point of your X and Z coordinates. In this example we will be using X1.5 and Z0.
   - "Ø" This is the diameter that you want to turn the part down to. In this example we will use -0.25.
   - Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
3. Define your machining conditions:

- You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using CSS and you will program the surface speed and the screen will change "S" to "VCC".
- SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
- Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
- Coolant on or off. Use the half-key to select.

4. Define your Roughing Pass:
   - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
   - VCC- This is your surface speed. In this example we are using a value of 300.
   - T- Enter is your tool number. In this example we are using T1.
   - D- Enter in your Tool Offset Number. In this example we are using D1.
5. Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.01.

SIMULATION

Before the execution of any part program, running your program in simulation is strongly recommended. The following steps will guide you on how to set-up your graphics.

1. Select the Graphics Icon from your canned cycle page.
2. Select F1- Type of Graphics
3. Select F4- Solid Graphics
4. Next step is to define your part size.
5. Press F2- Display Area.
6. For this example we will be using the following:

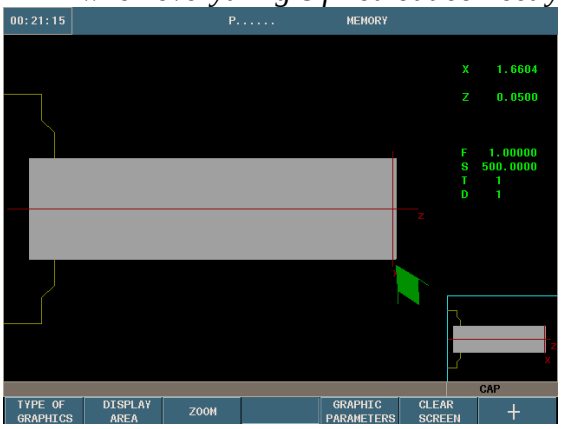
Z MIN
-5.000

Z MAX
0.05

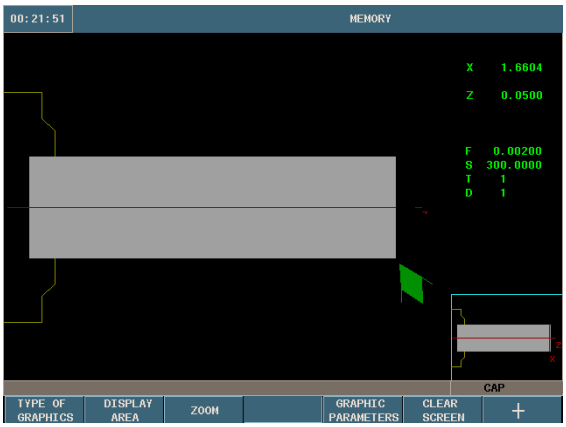
INSIDE D
0.000

OUTSIDE D
1.500

Note: Press the down arrow to get to the next box.
Press ENTER when everything's filled out correctly.



7. Now to run your program in simulation press the Cycle Start Button.
8. After simulating the cycle the following screen should appear:



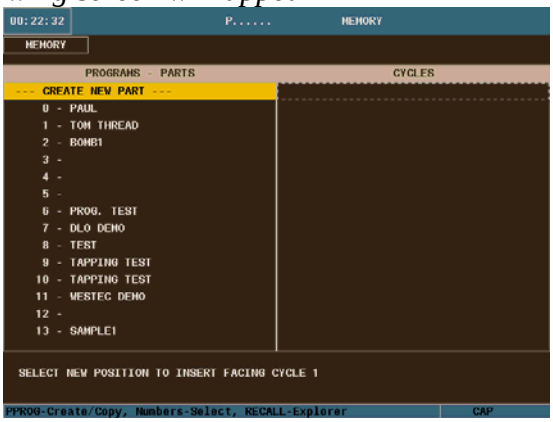
Note: If you notice no change in your graphics while running in simulation, go through step 6 again.

9. You are now ready to save this cycle to your Part-Program.

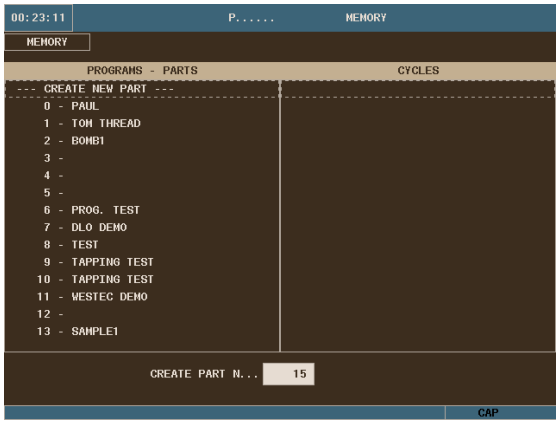
Storing Part Programs

1. Select The Part-Program Key twice.

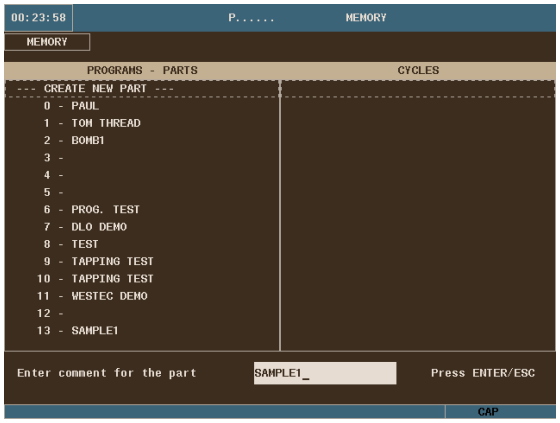
   - *The following screen will appear*

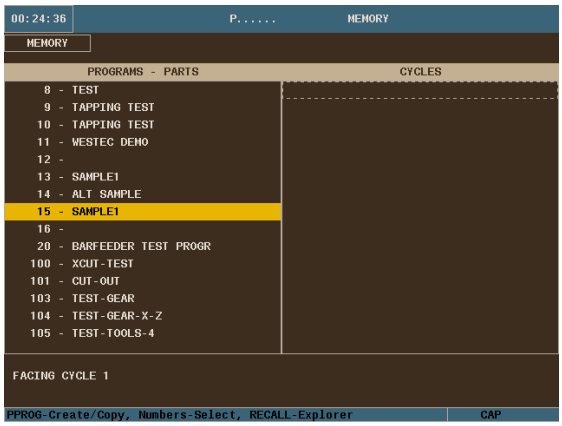


2. Highlight CREATE NEW PART and press the Part-Program Key once more. Enter a part number and press Enter. In this example we will us 15.
   - *The CNC will call up the first available number.*

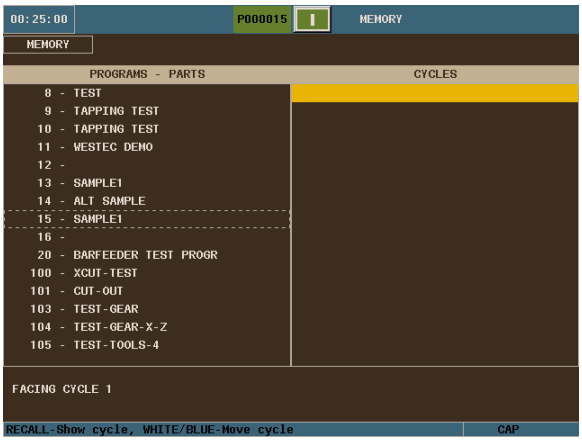


3. Enter a comment for your part-program and hit Enter. In this example we are using SAMPLE1.

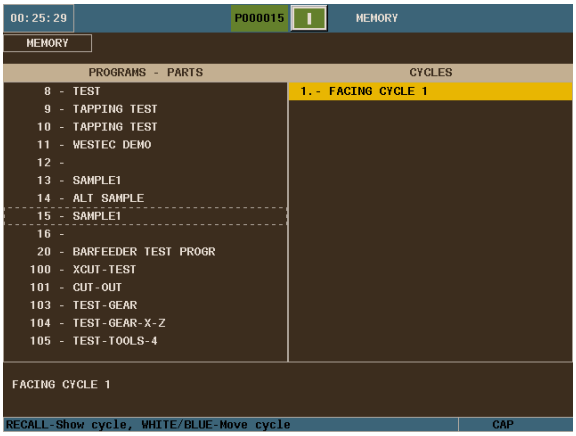




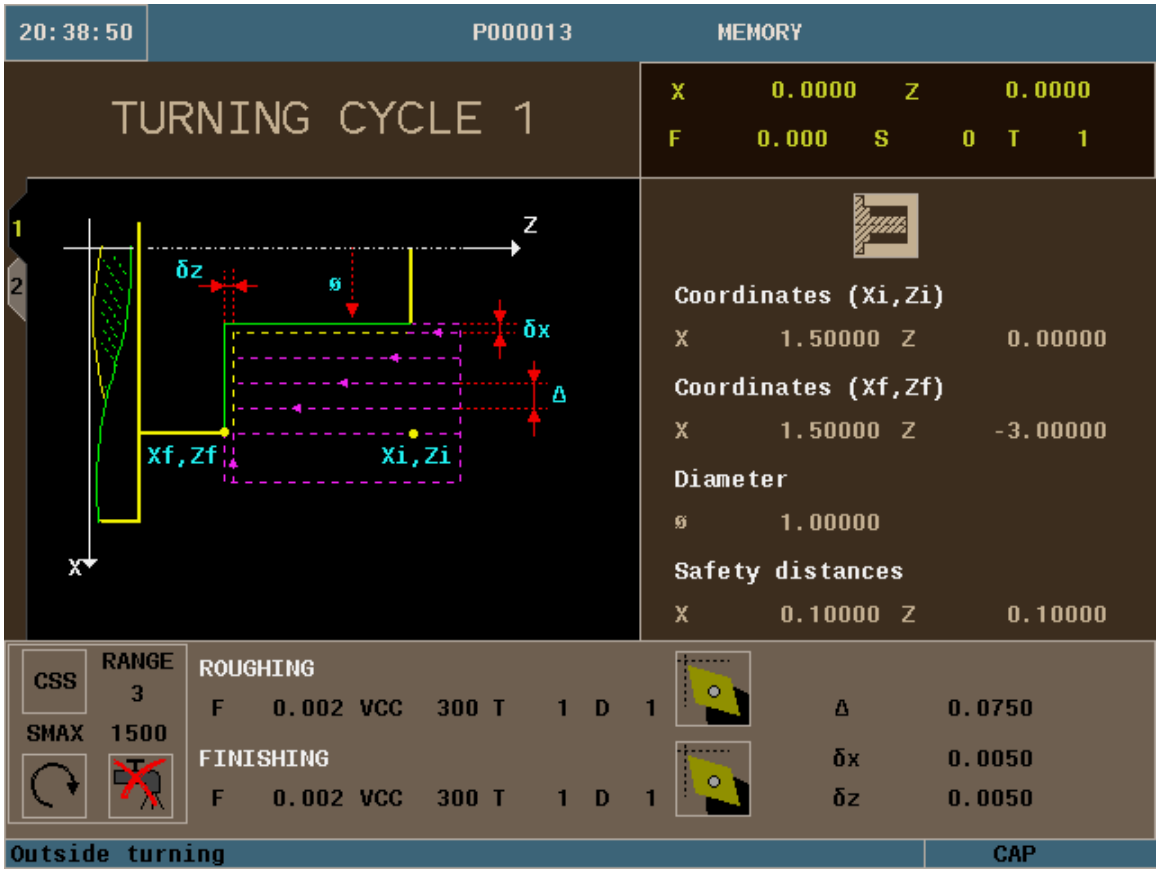
4. Select the Right Arrow Cursor. Now the right-hand column (cycles) will be highlighted.



5. Once this section is highlighted you may now press Enter and your cycle will be saved.



TURNING CYCLE



1. Select the F2 Icon.
2. For this operation we will be using Outside Turning. You can use the half-key to toggle from inside to outside turning when the icon is highlighted red.

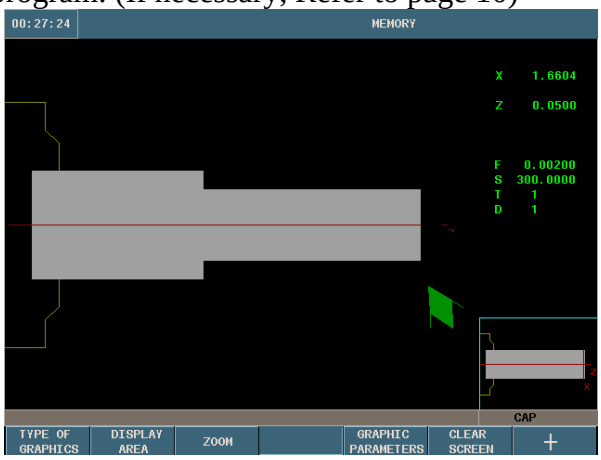
   Half-key

3. Define your part:
   - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1.5 and Z0.
   - (Xf, Zf)- This is your ending point of your X and Z coordinates. In this example we will be using X1.5 and Z-3.
   - "Ø" This is the diameter that you want to turn the part down to. In this example we will use 1.
   - Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.

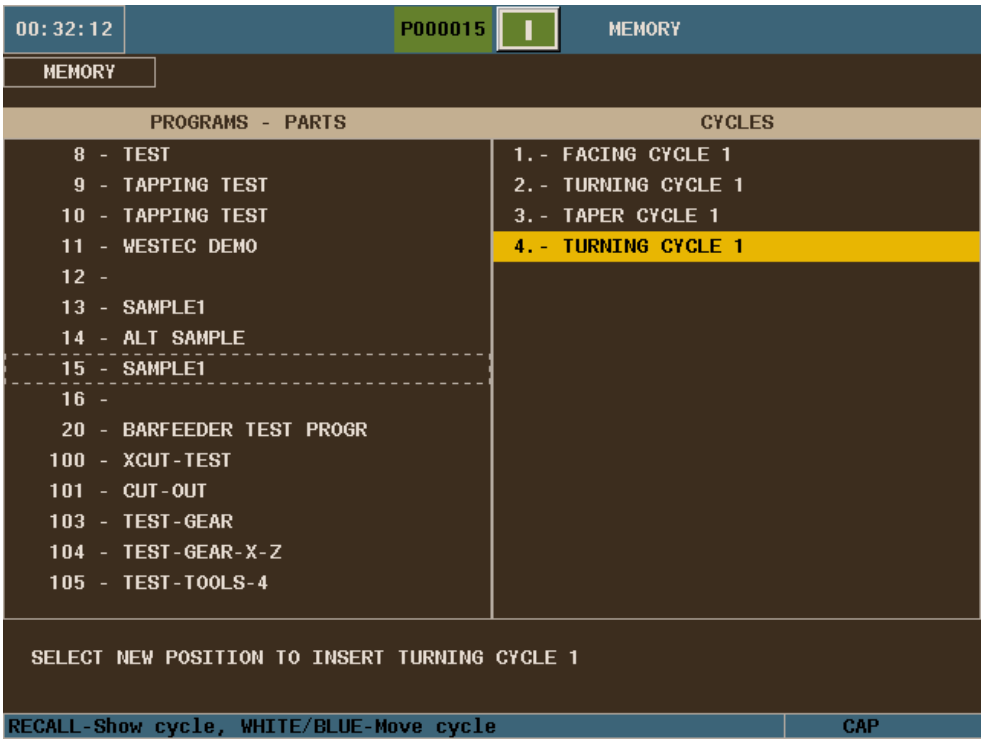
4. Define your machining conditions:
    - You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using CSS.
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

5. Define your Roughing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.
6. Define your Finishing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - δx - This is the amount of finishing stock that is going to be left on x for the finishing pass. For this example we will be using 0.005.
    - δz - This is the amount of finishing stock that is going to be left on z for the finishing pass. For this example we will be using 0.005.

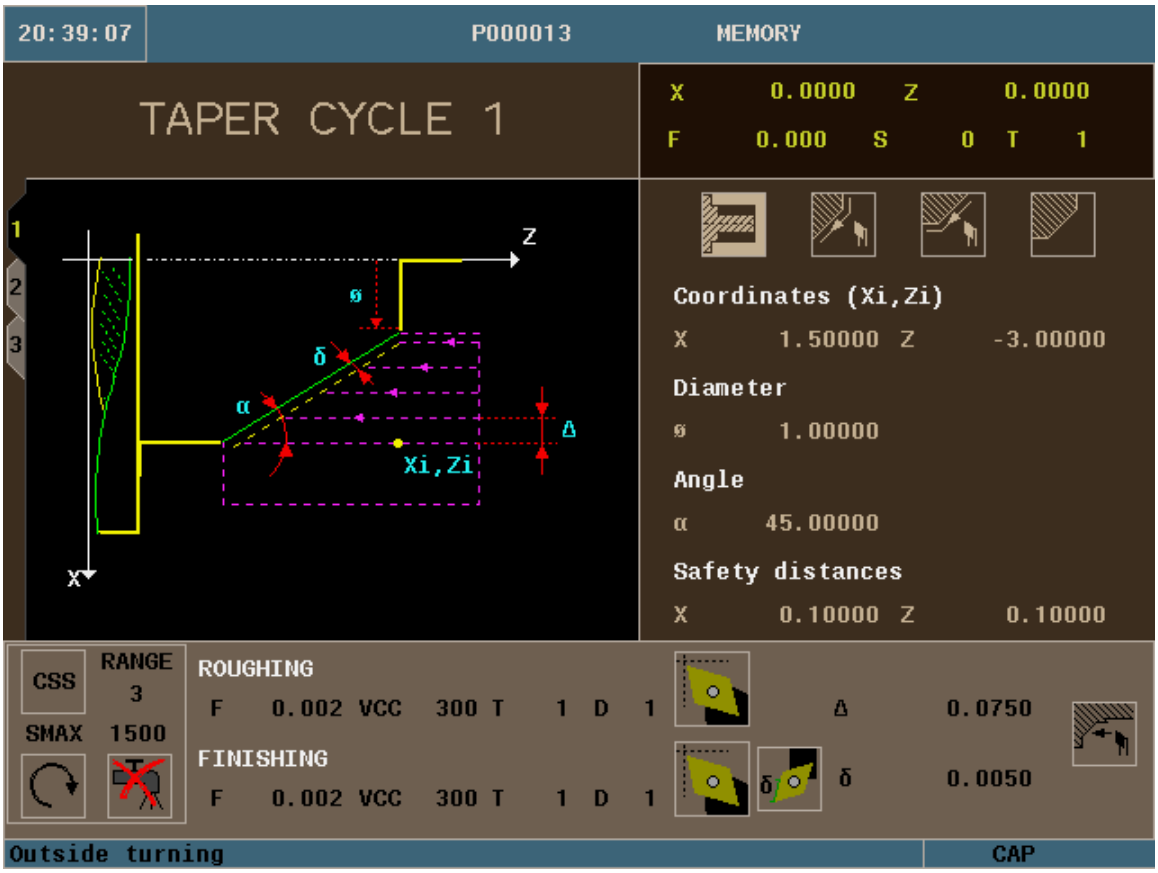
6. Simulate the program: (If necessary, Refer to page 10)



7. Save your part-program.



TAPER CYCLE



1. Select the F4 Icon.
2. For this operation we will be using Outside Turning. You can use the half-key to toggle from inside to outside turning when the icon is highlighted red.

   Half-key

3. Vertical or Horizontal entry of compensation. Use the half-key to toggle. For this example we will be using vertical.
4. Vertical or Horizontal exit of compensation. Use the half-key to toggle. For this example we will be using horizontal.
5. The turning area that we will be working in is Z+ quadrant.
6. Define your part:

- (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1.5 and Z-3.
- "Ø" This is the diameter that you want to turn the part down to. In this example we will use 1.
- α - This is the angle the taper will be on. In the example we will be using 45 degrees.
- Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.

7. Define your machining conditions:
    - You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using CSS.
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 2000.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

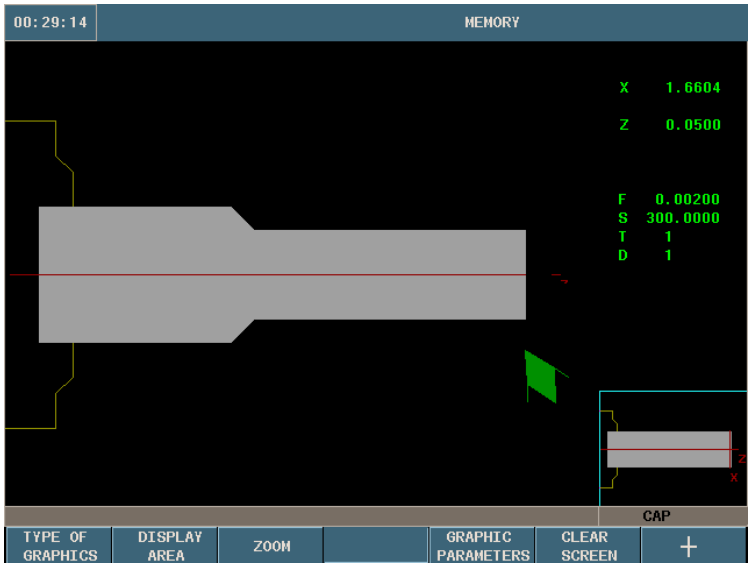
8. Define your Roughing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - S- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.

9. Define your Finishing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - S- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). For this example we will be using defining just 1.

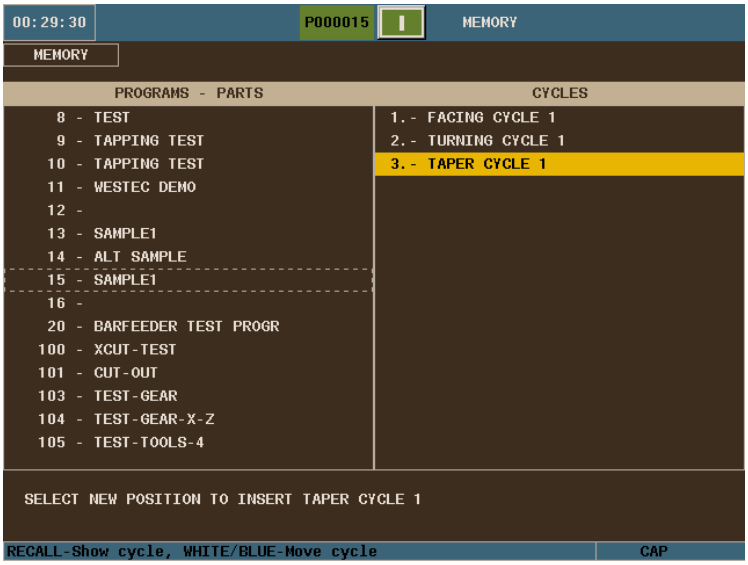
    - δ – This is the amount of finishing stock that is going to be left on for the finishing pass. For this example we will be using 0.05.

    - You can select to either machine along the Z axis or X axis. For this example we will use along Z.

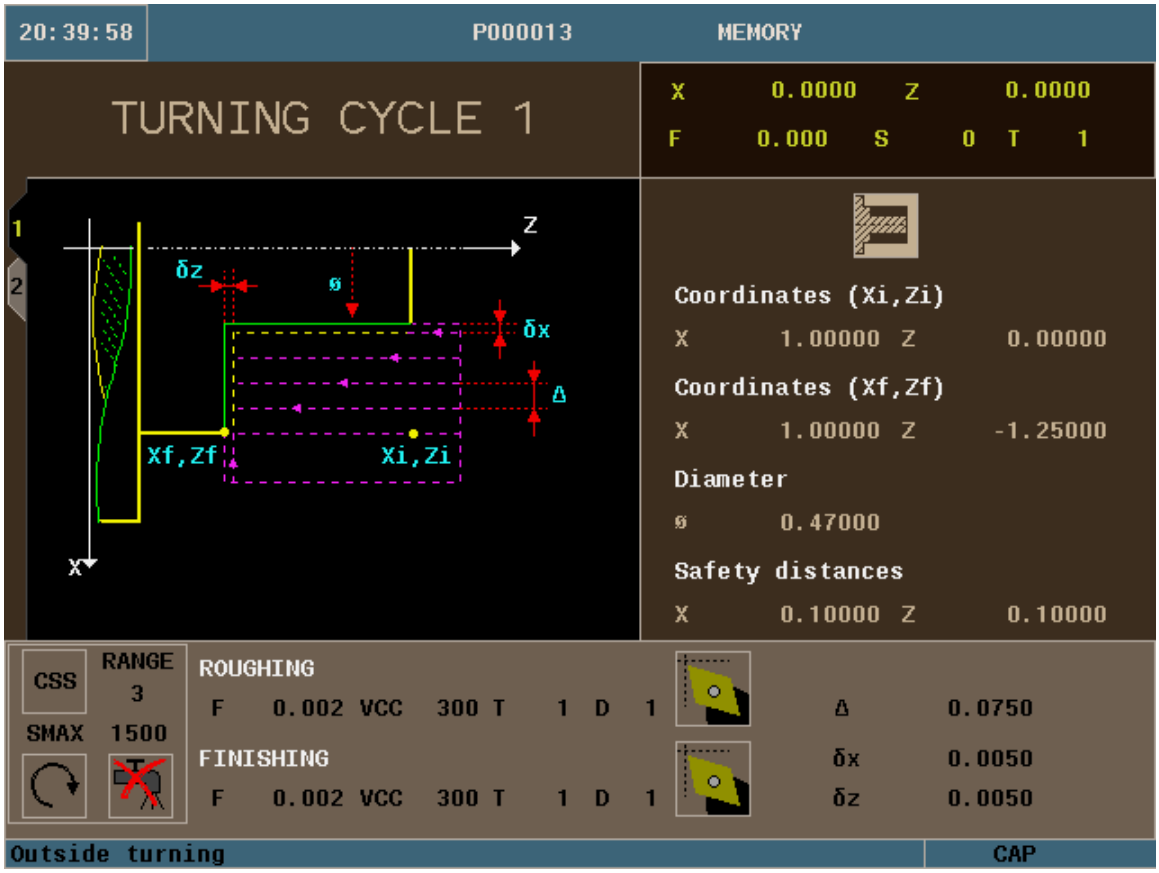
10. Simulate the program:



11. Save your Cycle to your part-program:



TURNING CYCLE



7. Select the F2 Icon.

8. For this operation we will be using Outside Turning. You can use the half-key to toggle from inside to outside turning when the icon is highlighted red.

   Half-key

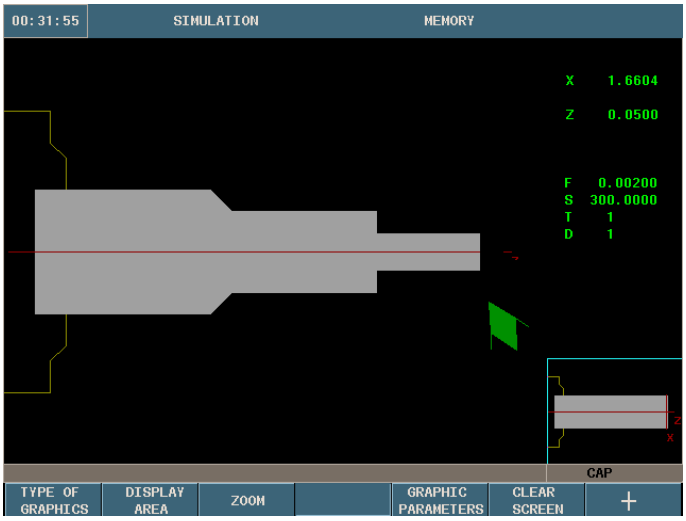
9. Define your part:
   - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1. and Z0.

- (Xf, Zf)- This is your ending point of your X and Z coordinates. In this example we will be using X1. and Z-1.25
- "Ø" This is the diameter that you want to turn the part down to. In this example we will use .48 to undercut for the thread
- Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.

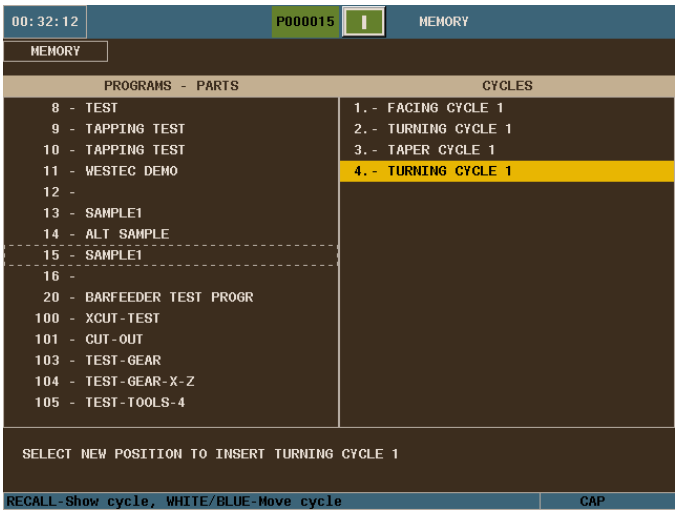
10. Define your machining conditions:
    - You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using CSS.
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

11. Define your Roughing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.
12. Define your Finishing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - δx - This is the amount of finishing stock that is going to be left on x for the finishing pass. For this example we will be using 0.005.
    - δz - This is the amount of finishing stock that is going to be left on z for the finishing pass. For this example we will be using 0.005.

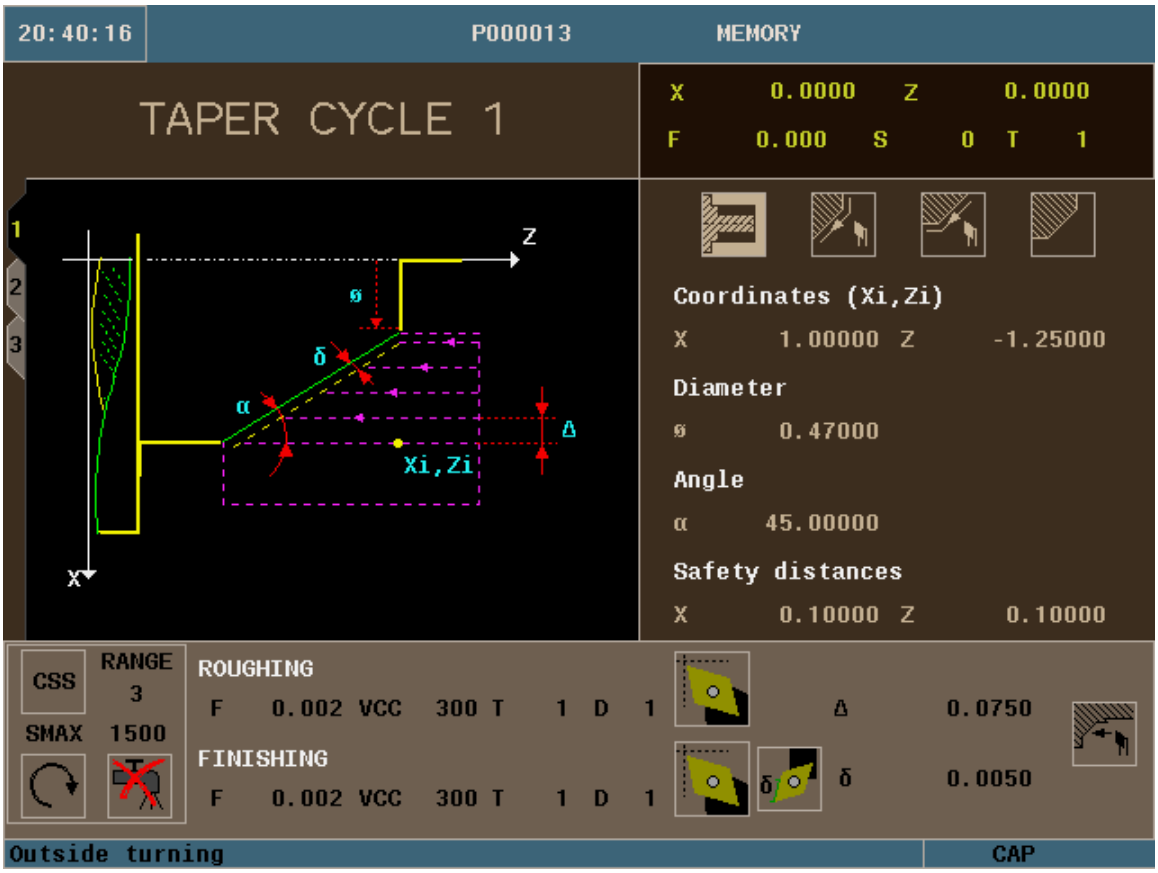
8. Simulate the program: (Refer to page 10)



9. Save your part-program. (Refer to page 11)



TAPER CYCLE



12. Select the F4 Icon.

13. For this operation we will be using Outside Turning. You can use the half-key to toggle from inside to outside turning when the icon is highlighted red.

    Half-key

14. Vertical or Horizontal entry of compensation. Use the half-key to toggle. For this example we will be using vertical.

15. Vertical or Horizontal exit of compensation. Use the half-key to toggle. For this example we will be using horizontal.

16. The turning area that we will be working in is Z+ quadrant.
17. Define your part:

- (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1. and Z-1.25
- "Ø" This is the diameter that you want to turn the part down to. In this example we will use .48
- α - This is the angle the taper will be on. In the example we will be using 45 degrees.
- Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.

18. Define your machining conditions:
    - You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using CSS.
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

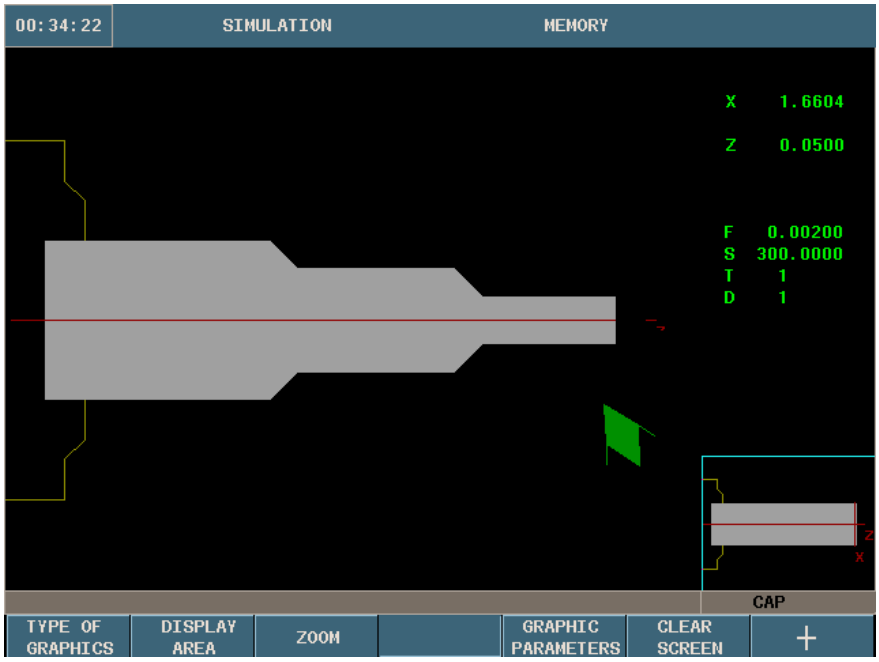
19. Define your Roughing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.005 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.

20. Define your Finishing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.005 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). For this example we will be defining just 1.

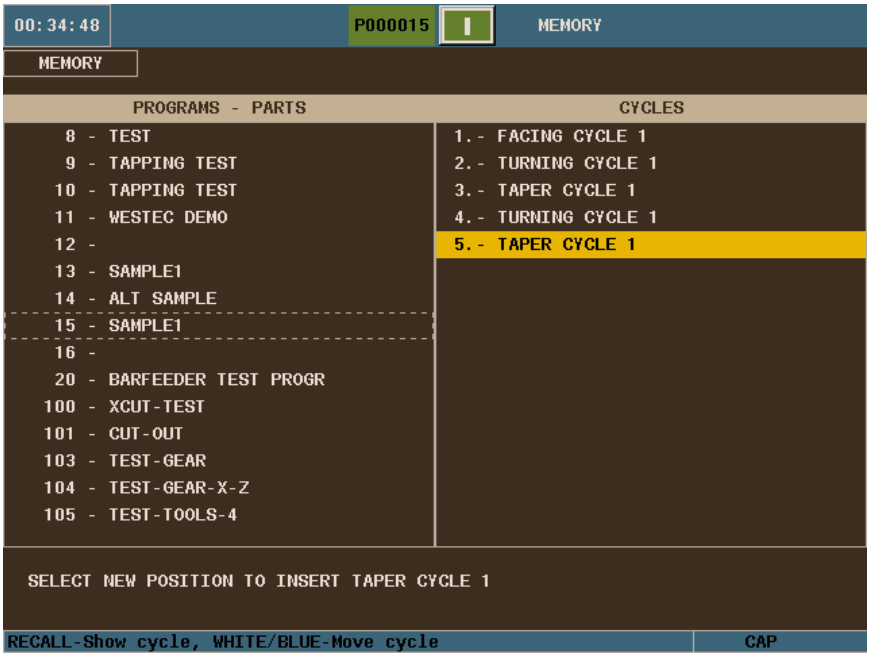
    - δ – This is the amount of finishing stock that is going to be left on for the finishing pass. For this example we will be using 0.05.

    - You can select to either machine along the Z axis or X axis. For this example we will use along Z.

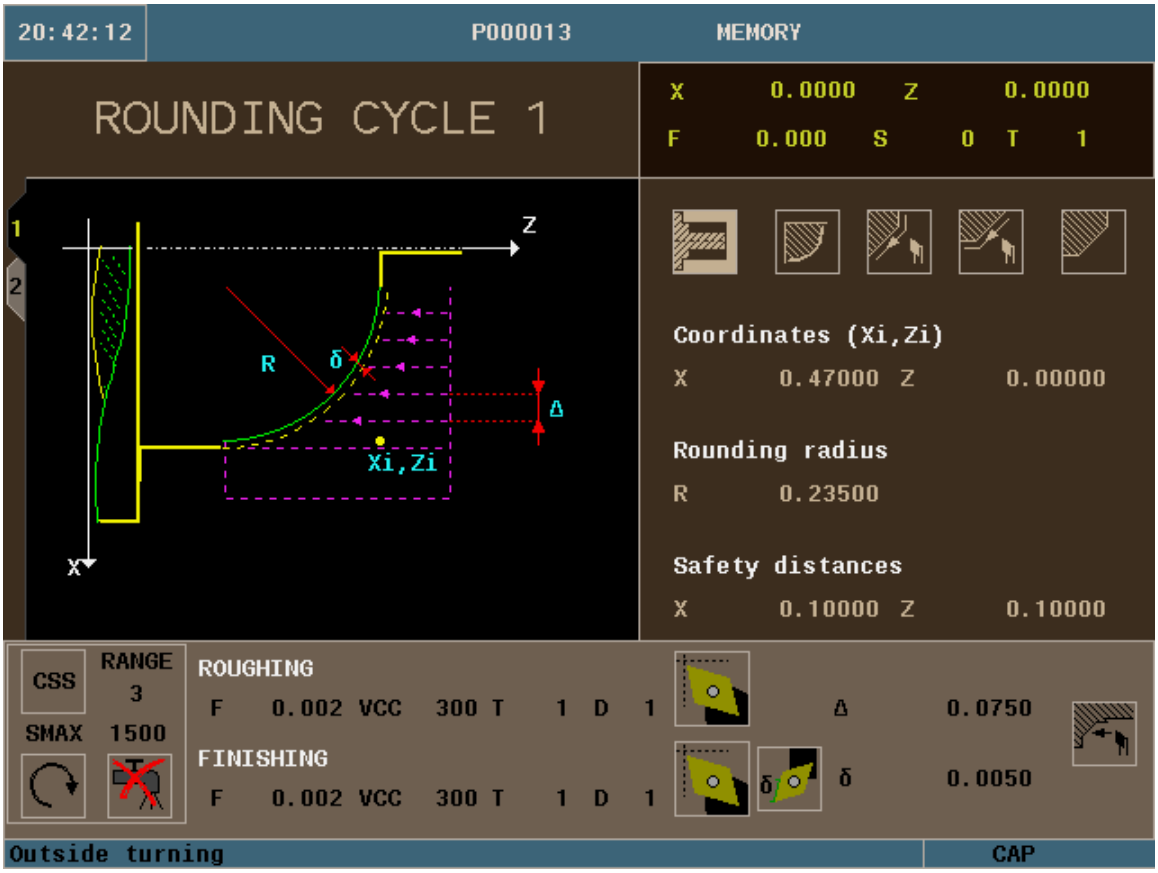
21. Simulate the program: (Refer to page 10)



22. Save your Cycle to your part-program: (Refer to page 11)



ROUNDING CYCLE



1. Select the F5 Icon.
2. For this operation we will be using Outside Turning. You can use the half-key to toggle from inside to outside turning when the icon is highlighted red.

   Half-key
3. You can choose either convex or concaved rounding. Use the half-key to toggle. We will be using convex rounding for this example.
4. Vertical or Horizontal entry of compensation. Use the half-key to toggle. For this example we will be using vertical.
5. Vertical or Horizontal exit of compensation. Use the half-key to toggle. For this example we will be using horizontal.

6. The turning area that we will be working in is Z+ quadrant.
7. Define your part:
    - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X.48 and Z0.
    - R- This is where you put your radius value. In this example we will be using .24
    - Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
8. Define your machining conditions:
    - You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using CSS.
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

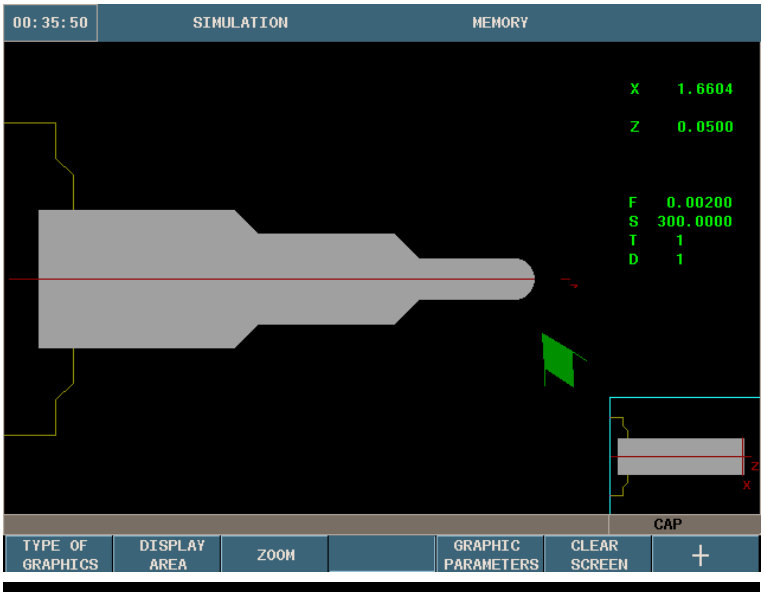
9. Define your Roughing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.

10. Define your Finishing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T1.
    - D- Enter in your Tool Offset Number. In this example we are using D1.
    - Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). For this example we will be using defining just 1.

    - δ – This is the amount of finishing stock that is going to be left on for the finishing pass. For this example we will be using 0.05.

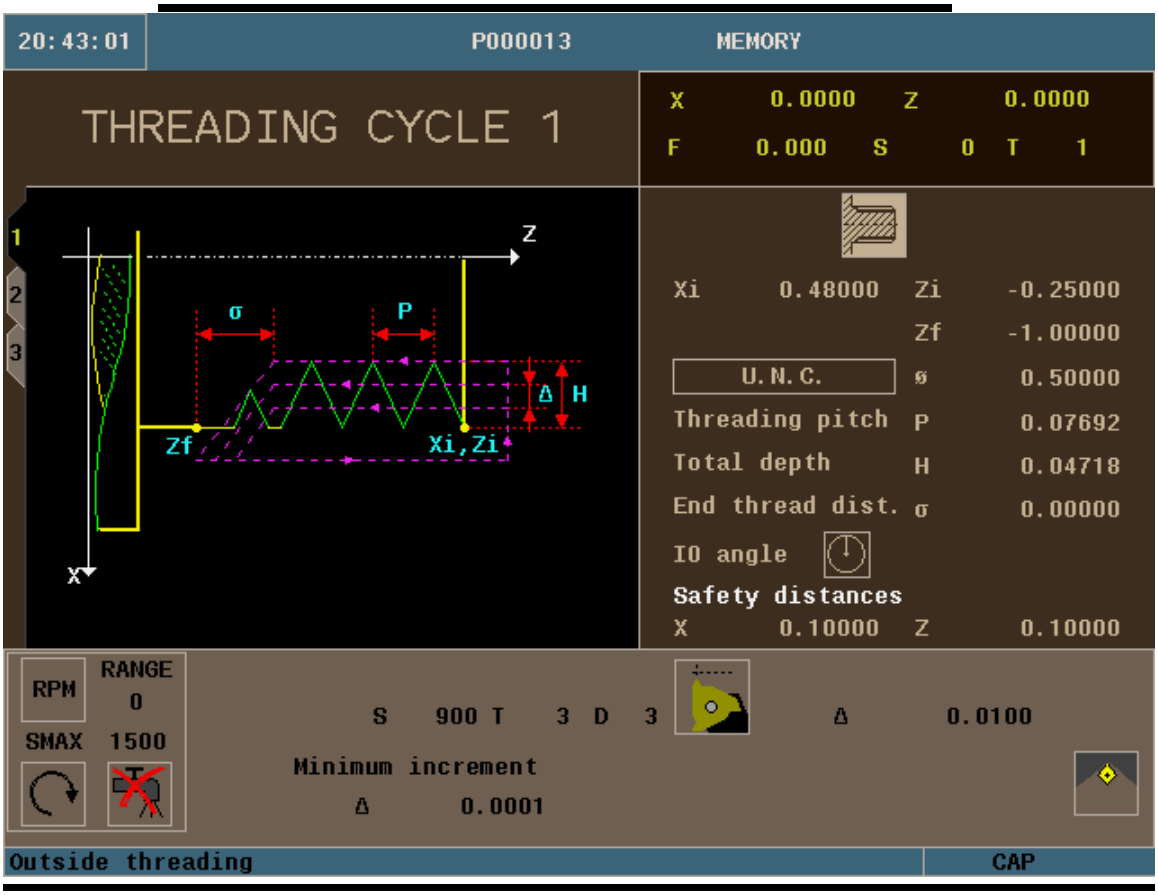
    - You can select to either machine along the Z axis or X axis. For this example we will use along Z.

11. Simulate the program: (Refer to page 10)



12. Save your Cycle to your part-program: (Refer to page 11)

THREADING CYCLE



1. Select the F6 Icon

2. For this operation we will be using Outside Threading. You can use the half-key to toggle from inside to outside threading when the icon is highlighted red.

   Half-key

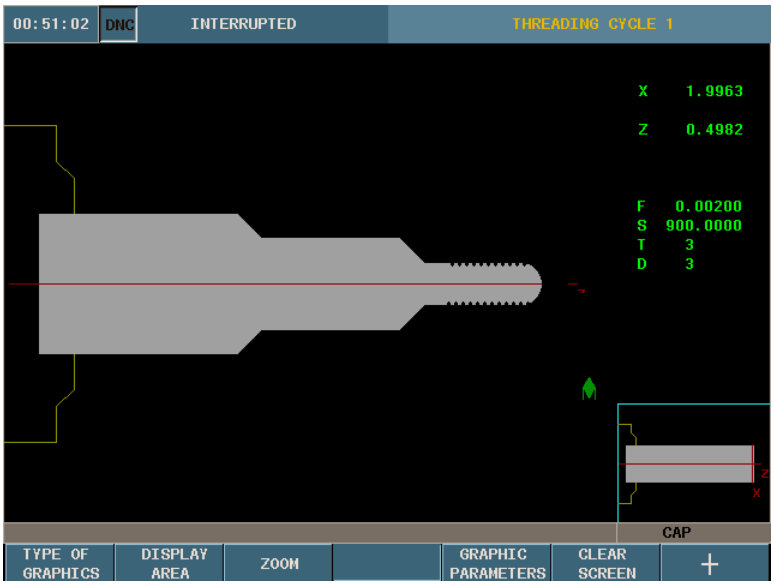
3. Define your part:
   - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X.5 and Z-.25
   - (Zf)- This is your ending point of your Z coordinate. In this example we will be using Z-1.
   - Select the thread type that you would like to use. You can use the half-key to toggle through the different types of thread. In this example we will be using U.N.C.
   - "Ø" This is the diameter of the material that is being threaded. We will be using .5 for our example.

- σ - This indicates at what distance from the end of the thread the tool starts exiting. In this example we will be using 0.
- IO angle- you can program with or without entry angle. For this example we will program without.
- Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.

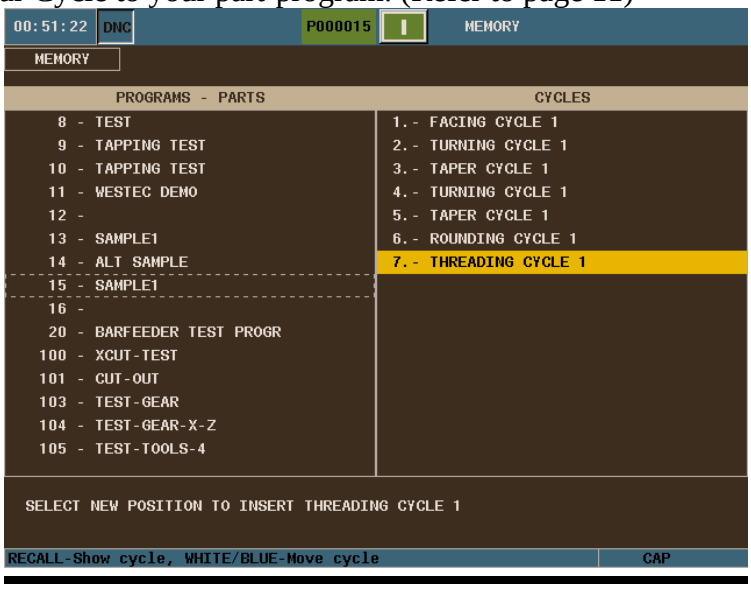
4. Define your machining conditions:
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

5. Define your Passes:
    - S- This is your spindle speed. In this example we are using a value of 900 rpm.
    - T- Enter is your tool number. In this example we are using T3.
    - D- Enter in your Tool Offset Number. In this example we are using D3.
    - Max Δ- This is the maximum amount of material that is going to be taken off per pass. In this example we will be using 0.01.
    - Min Δ- This is the minimal amount of material that is going to be taken off per pass. In this example we will be using 0.001.
    - Repeat last threading pass or not. We will not be repeating last pass for this example.

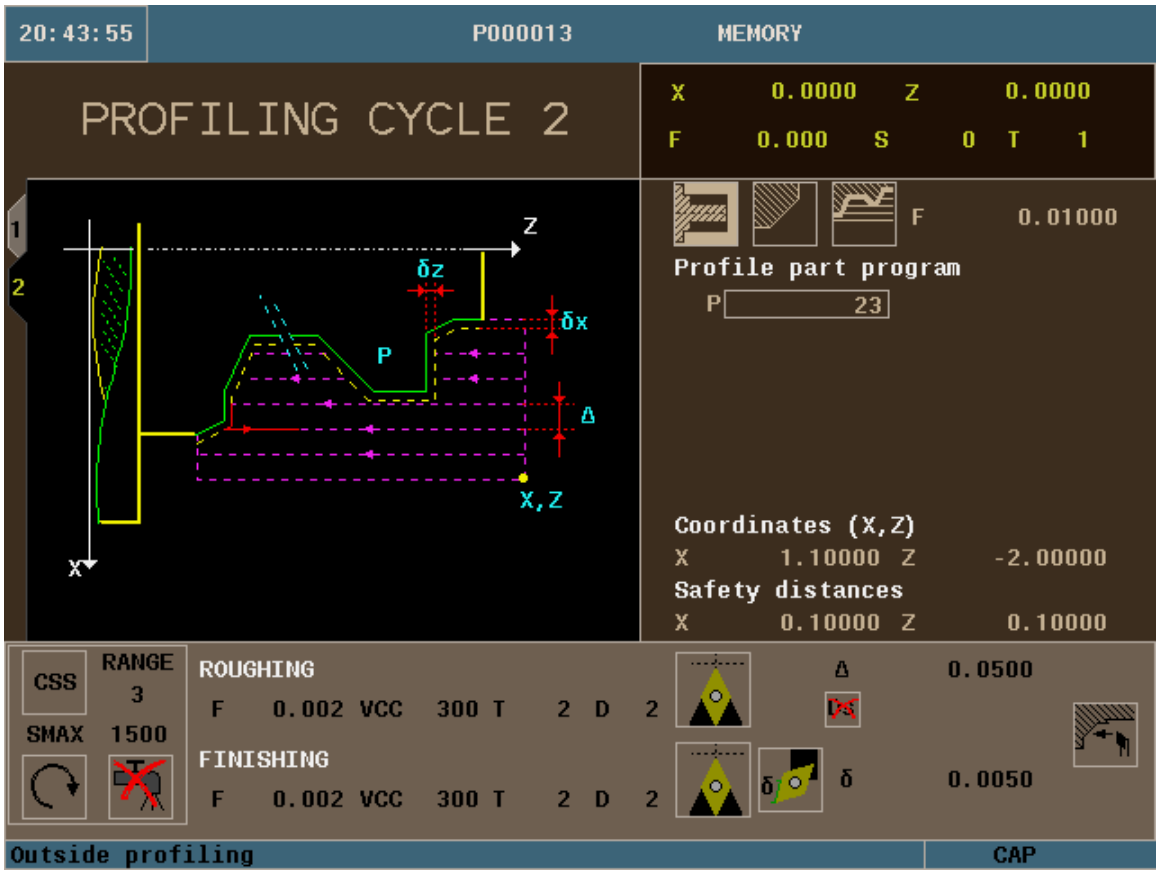
6. Simulate the program: (Refer to page 10)



7. Save your Cycle to your part-program: (Refer to page 11)



PROFILING CYCLE



1. Select the Profiling Icon

2. For this operation we will be using Outside Profiling. You can use the half-key to toggle from inside to outside profiling when the icon is highlighted red.

Half-key

3. The turning area that we will be working in is Z+ quadrant.

4. You can either machine your profile in 2 ways: paraxial machining (1 axis at a time) and machining following the profile. For this example we will be following the profile.

5. E – How far away from the profile we will be starting. For this example we will use 0.10.

6. P – This is going to be our new profile number. In this example put in number 23 and then press RECALL. You will now be on the mini-cad page. Follow the following steps for our demo part:

    - Press Edit (F1)
    - Press Profile (F1)
    - Insert the initial points on the right hand side. → Z1: -2, X1: 1
    - Press Validate (F7)

    - Press Straight Line (F1)
    - Insert the following: → Z2: -2.25, X2: .5
    - Press Validate (F7)

    - Press Clockwise Arc (F2)
    - Insert the following: → Z2: -2. 5, X2: 1., Zc: **LEAVE BLANK**, Xc: **LEAVE BLANK**, R: 0.25
    - Press Validate (F7)
    - Press ENTER

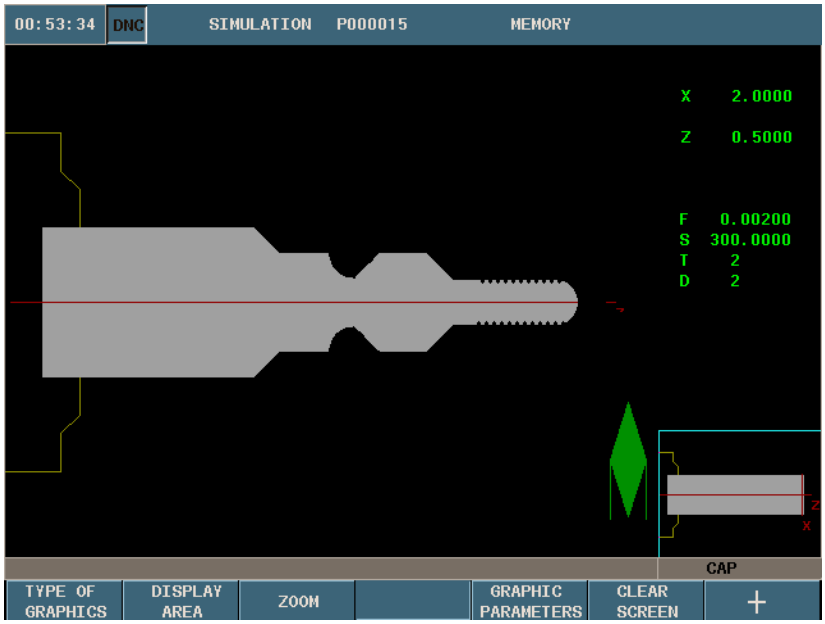
    - Press Save & Continue (F7)
    - Press ESC 2 times.
    - Press Finish (F5)
    - Enter in a comment. (DEMO)
    - Press ENTER.
7. Once you are back on the profiling cycle 2 screen press Enter.
8. (X,Z) – coordinates of the starting points. For this example we will be using X: 1.1 , Z: -2.
9. Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
10. Define your machining conditions:
    - You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using CSS.
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

11. Define your Roughing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.02 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T2.
    - D- Enter in your Tool Offset Number. In this example we are using D2.

- Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.05.

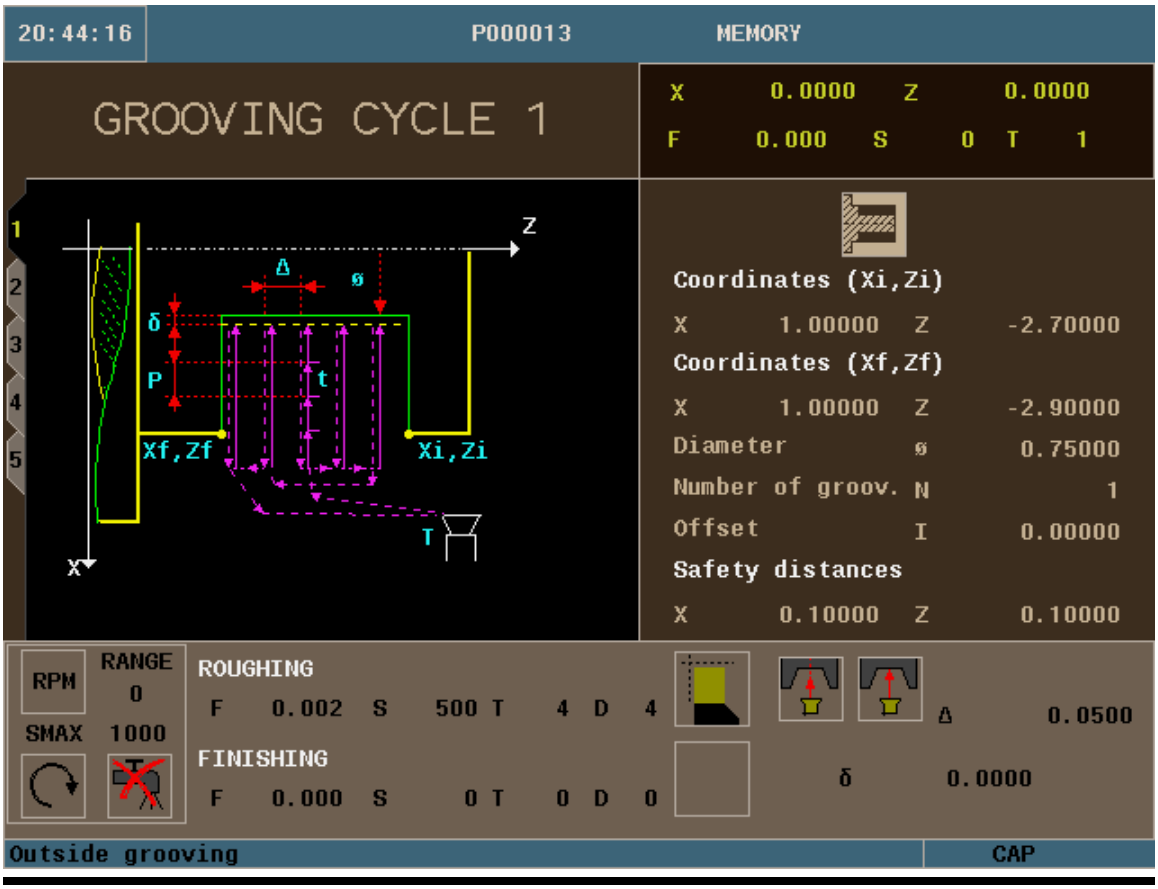
12. Define your Finishing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.015 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T2.
    - D- Enter in your Tool Offset Number. In this example we are using D2.
    - δ – This is the amount of finishing stock that is going to be left on for the finishing pass. For this example we will be using 0.05.
    - You can select to either machine along the Z axis or X axis. For this example we will use along Z.

8. Simulate the program: (Refer to page 10)



9. Save your Cycle to your part-program: (Refer to page 11)

GROOVING CYCLE



1. Select the F7 Icon

2. For this operation we will be using Outside Grooving. You can use the half-key to toggle from inside to outside grooving when the icon is highlighted red.

   Half-key

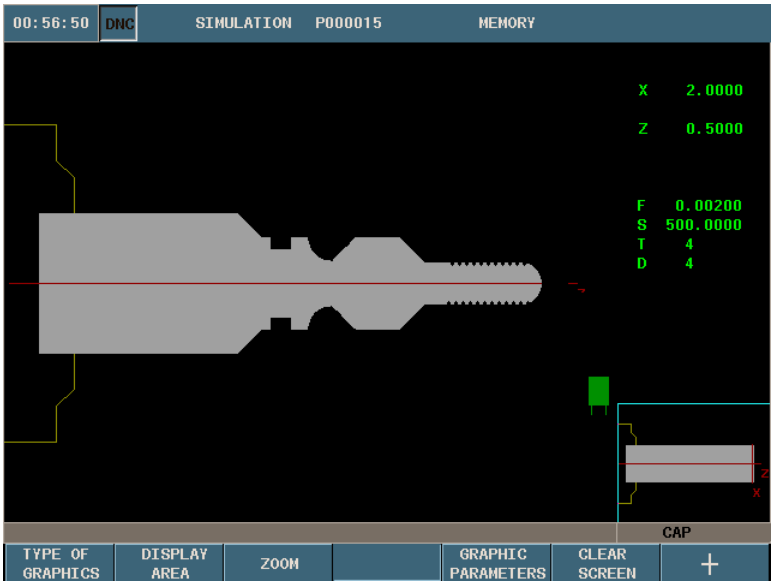
3. Define your part:
   - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1.5 and Z-2.7
   - (Xf, Zf)- This is your ending point of your X and Z coordinates. In this example we will be using X1.5 and Z-2.9
   - "Ø" This is the diameter that you want to turn the groove down to. In this example we will use .75.
   - N- Number of grooves. We will only be making 1 groove in this example.

- I- This is the offset in between each groove. We will put it at 0 since we are only making 1 grove.
- Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.

4. Define your machining conditions:
    - You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using CSS.
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

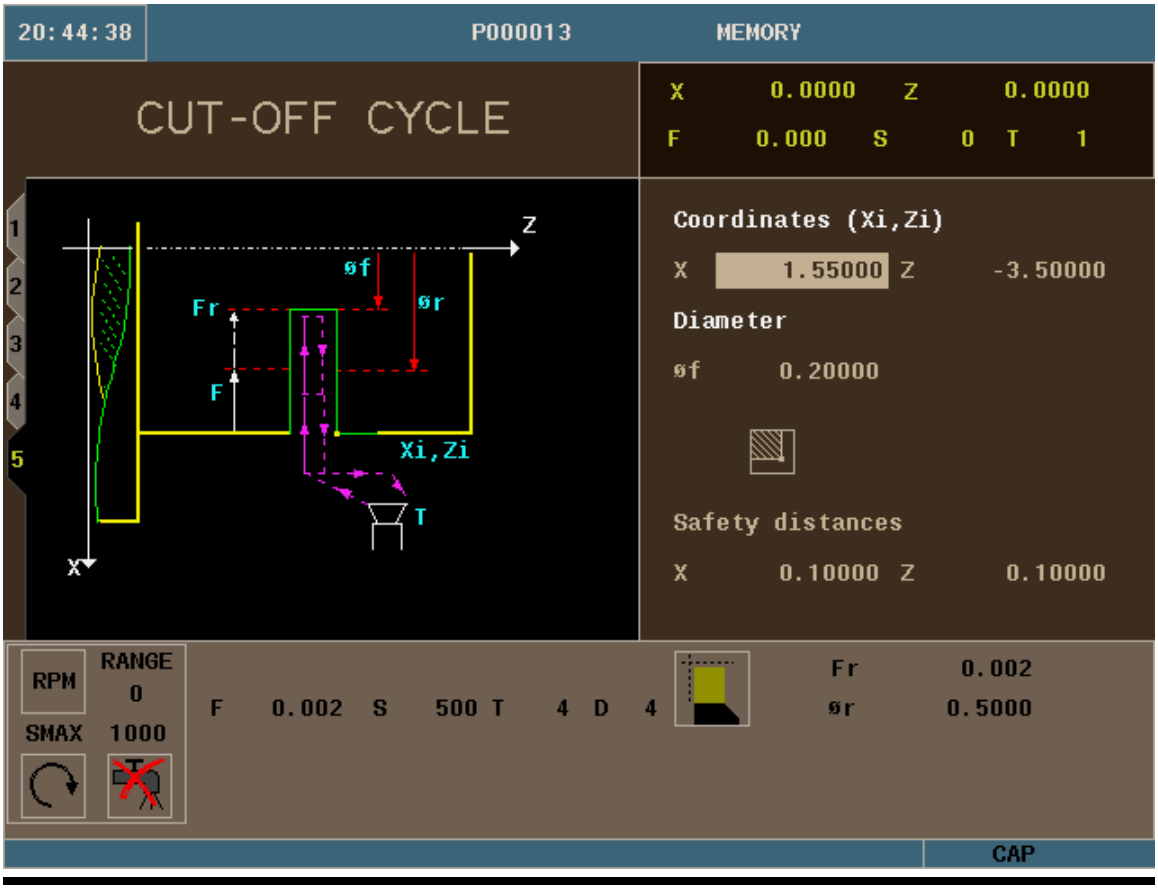
5. Define your Roughing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - VCC- This is your surface speed. In this example we are using a value of 300.
    - T- Enter is your tool number. In this example we are using T4.
    - D- Enter in your Tool Offset Number. In this example we are using D4.
    - This symbol represents if you want to machine from the center of the slot or from the starting point. For this example we will be starting in the center of the slot.
    - This symbol means with or without penetration chip removal. We will use penetration without chip removal for this example.
    - Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.05.

2. Simulate the program: (Refer to page 10)



3. Save your Cycle to your part-program: (Refer to page 11)

CUT-OFF CYCLE



6. Select the F7 Icon

7. Press the Level Cycle key.

Press this key until you are in CUT-OFF CYCLE.

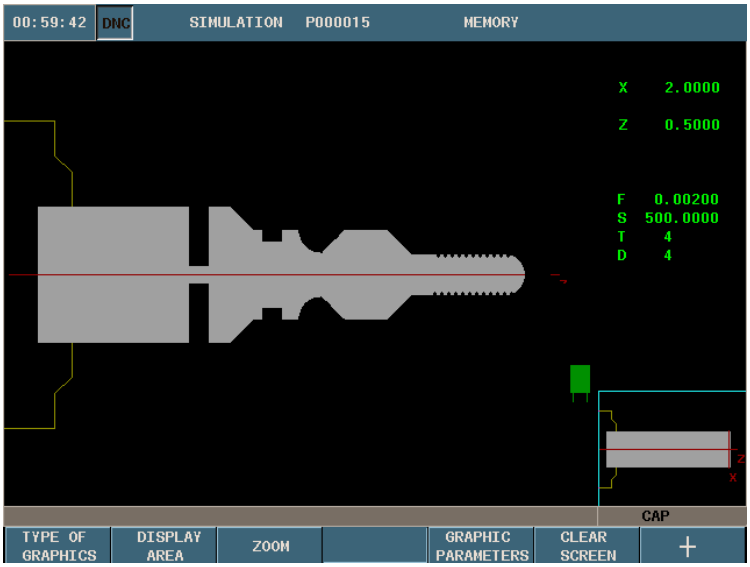
8. Define your part:
   - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1.55 and Z-3.5
   - "Øf" This is the diameter that you want to turn the groove down to. In this example we will use .2.

- Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.

9. Define your machining conditions:
    - You can use either RPM or CSS when machining. Use the half-key to toggle from one to the other. In this example we will be using RPM.
    - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
    - Spindle direction. The half-key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
    - Coolant on or off. Use the half-key to select.

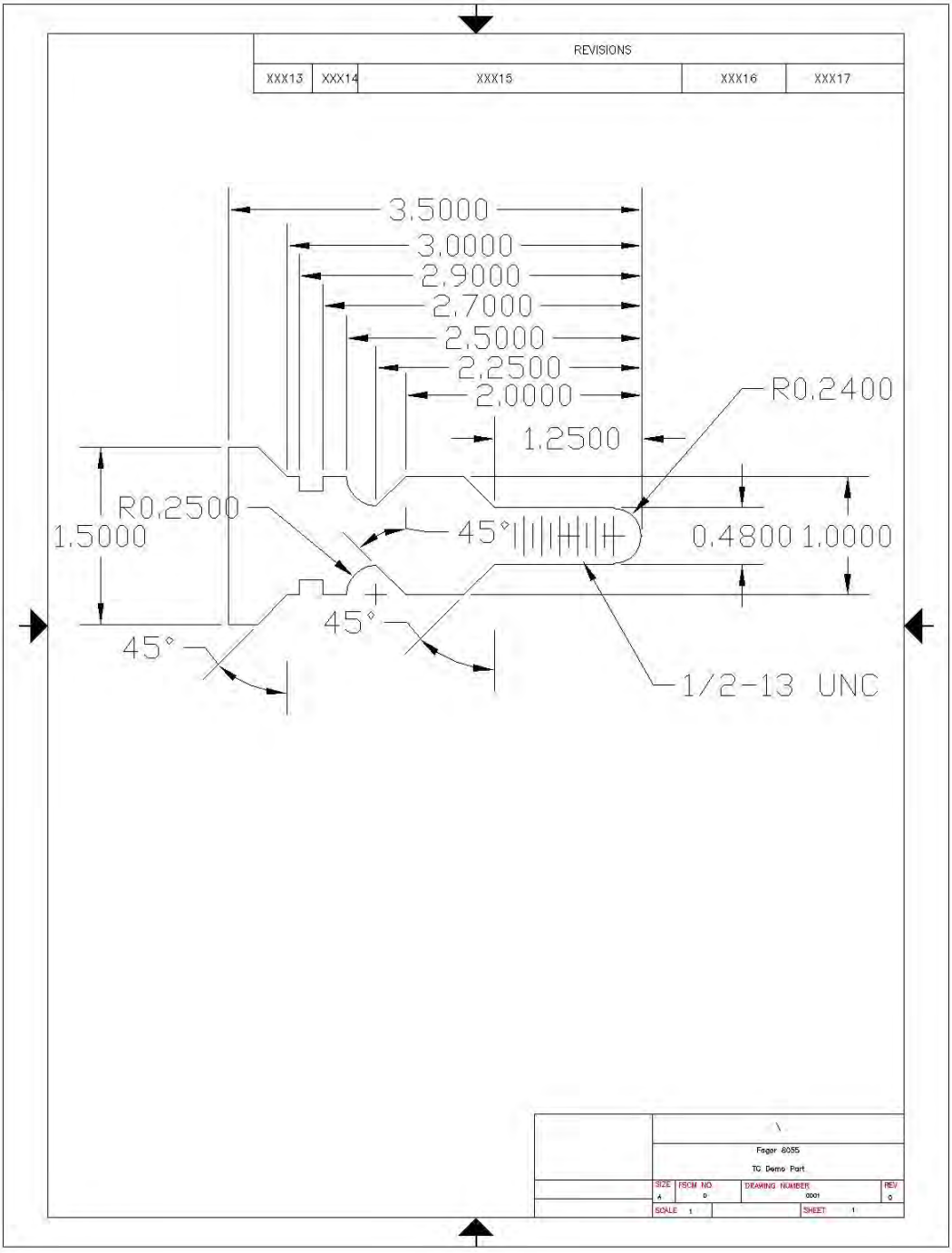
10. Define your Roughing Pass:
    - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
    - S- This is your Spindle speed. In this example we are using a value of 500.
    - T- Enter is your tool number. In this example we are using T4.
    - D- Enter in your Tool Offset Number. In this example we are using D4.
    - "Fr"- is your Final slow feedrate. How much material you are going to take off per pass. In this example we will be using 0.002.
    - "Ør"- This is your Gear ratio diameter for the feedrate. In this example we will be using 0.5.

4. Simulate the program: (Refer to page 10)



5. Save your Cycle to your part-program: (Refer to page 11)

You have now created and simulated your part-program and are ready for execution.

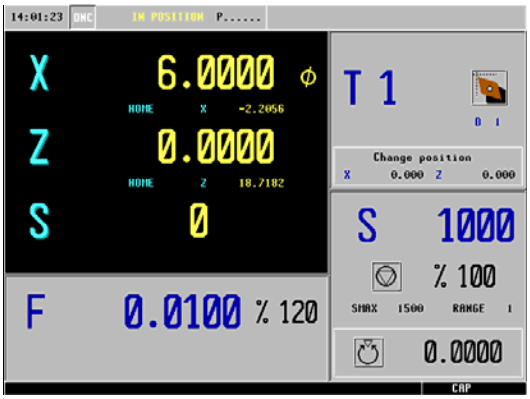

REVISIONS
XXX13
XXX14
XXX15
XXX16
XXX17
3.5000
3.0000
2.9000
2.7000
2.5000
2.2500
2.0000
1.2500
R0.2400
R0.2500
1.5000
45°
0.4800
1.0000
45°
45°
1/2-13 UNC
Fagor 8055
TC Demo Part
SIZE
A
FSCM NO.
0
DRAWING NUMBER
0001
REV
0
SCALE 1
SHEET 1

Miscellaneous Features

ZERO OFFSETS

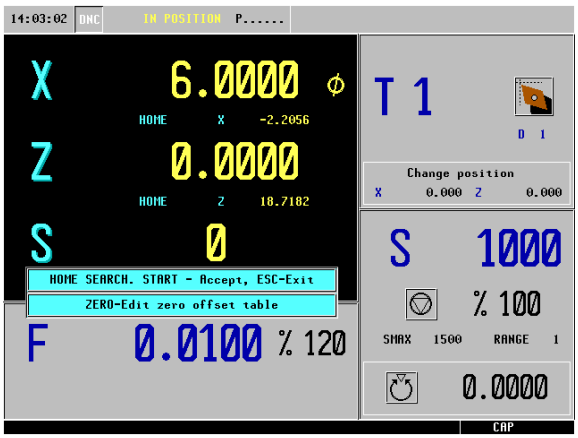
There are 20 absolute zero offsets. G54 through G159N20. This feature allows us to store a part zero position into a table. With this option you are able to recall your part zero position without having to touch off on your part again. This feature is useful when having parts that are commonly machined. To setup your Zero Offsets perform the following task:

1. From the Jog Screen setup your part zero position

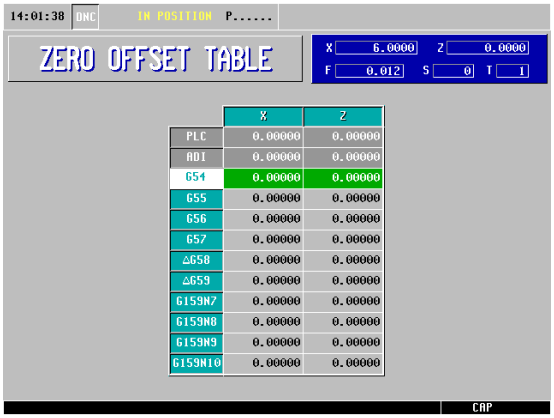


**Note: Refer to page 7 of the manual on how to preset your part zero*

2. Select the Zero Icon



3. Select the Zero Icon for a second time to enter the Zero Offset Table



** Note: All of the coordinate values for G54 are currently at zero*

4. Select the Recall Key and press ENTER.

** Note: The coordinates are now the exact value of their position away from home*

**Note: Your values will not be stored if you do not hit enter*

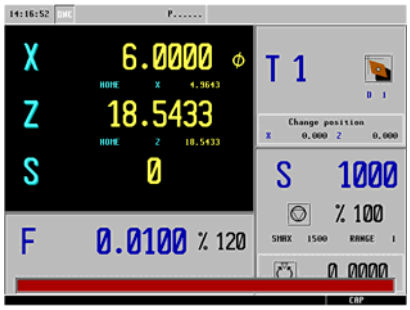
(To activate Zero Offsets, refer to page 46.)

ISO MODE

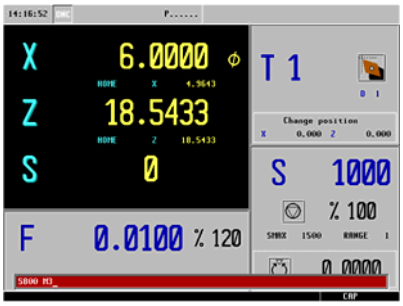
The ISO key gives you access to the MDI Mode. What this feature allows you to do is perform one or more functions at a time. It allows you to move more than one axes at a time, start the spindle, set the feed, insert G-Codes, etc. To operate in ISO Mode, execute the following instructions:

1. From the Jog Screen select the ISO key

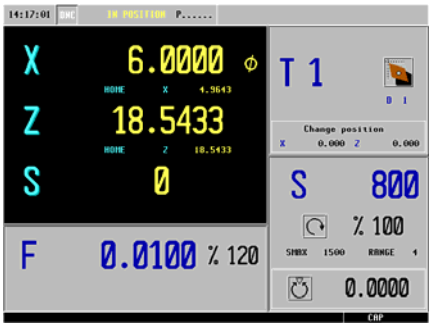
- In this example we will be turning on the spindle at a clockwise rotation at 800 rpm's
- The CNC will display a red window at the bottom of the screen



2. Type in S800 M3



3. Press Cycle Start

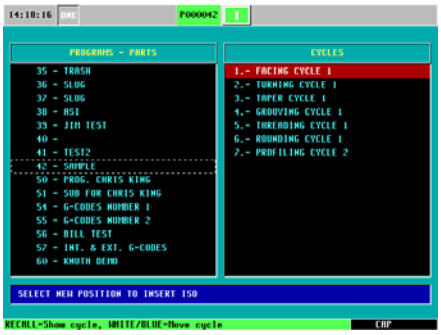


**Note: The spindle speed has changed to 100 and the icon has changed to a clock-wise position.*

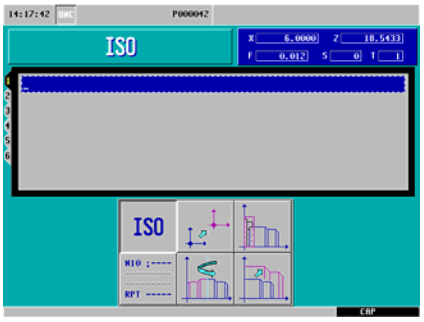
INSERTING CODE IN-BETWEEN CANNED CYCLES

This feature allow us to insert M and G-Codes in-between our cycles in our library. In this example we will be inserting a G54 to start our program and a M00 in-between each cycle. The M00 will stop the program from executing the next programmed cycle until we hit cycle start. This feature is helpful in examining the tool after the completion of each cycle.

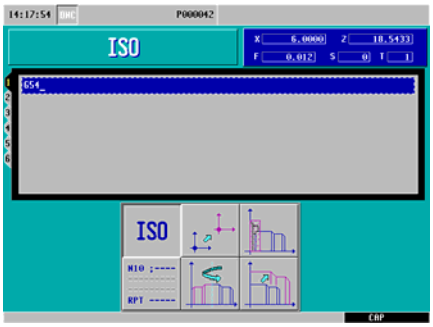
1. From the Jog Screen select the P.PROG key



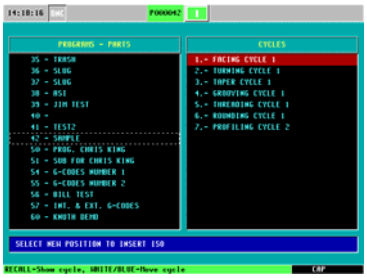
2. Select the ISO key



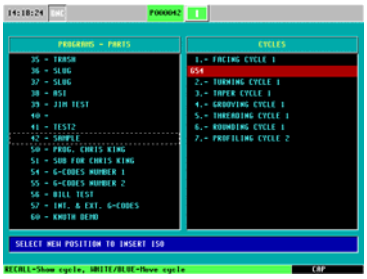
3. Type in G54



4. Select the P.PROG Key

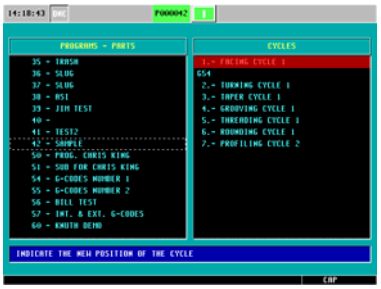


5. Press Enter



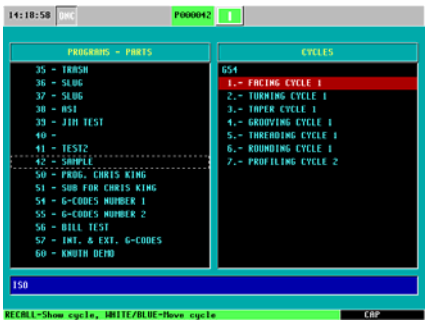
**Note: Make sure our first programmed cycle is highlighted in red before hitting Enter.*

6. Highlight our first programmed cycle and select the Half-Key



**Note: This allows us to insert our first programmed cycle underneath the G54.*

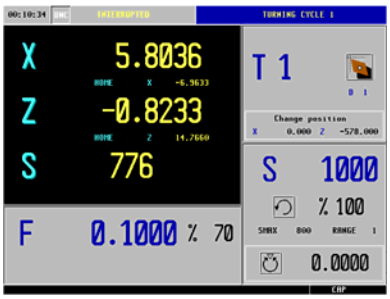
7. Arrow down to G54 and hit enter.



TOOL INSPECTION

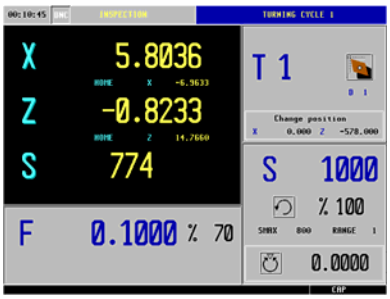
This feature allows us to examine our tool at any point while executing our program. Whether the tool breaks or if you are checking the status, this feature will allow you to stop your program, move your axes around and begin your program from the point you have left off at.

1. Press the STOP Key while the program is in execution



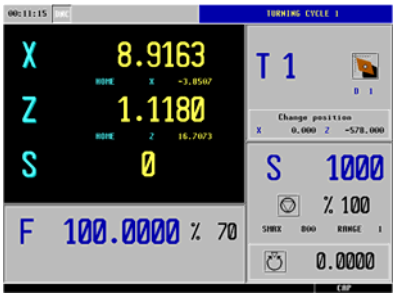
**Note: You will notice on the top of the screen it will read INTERRUPTED*

2. Press T (this is for Tool Inspection)



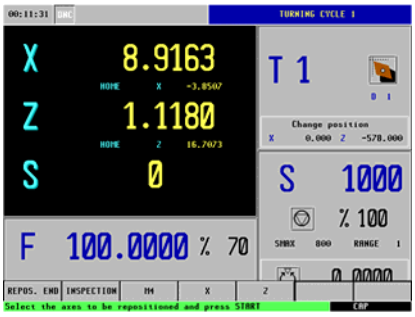
**Note: Notice the top of your screen now reads INSPECTION*

3. Execute the next step of commands:
   - Jog your X and Z Axis out of the way / Turn off your spindle.



**Note: Notice the coordinate values are different from when we stopped the program*

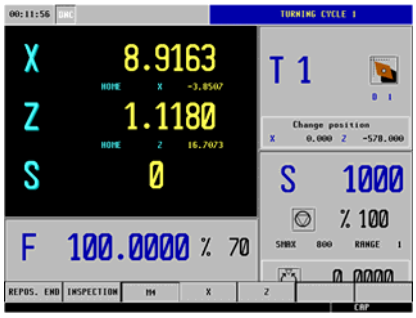
4. After examining the tool or changing it, press Cycle Start.



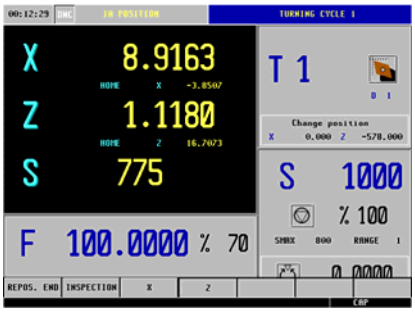
Note: After pressing Cycle Start the CNC will ask you to return to the last executed block.

Here is a recommended example:

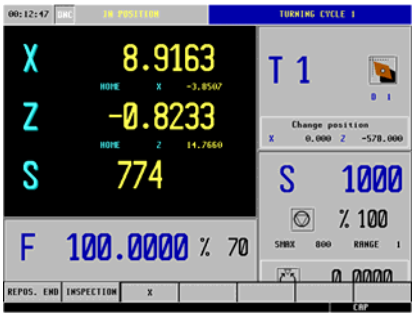
5. Select M4 and the Cycle Start to start your spindle.



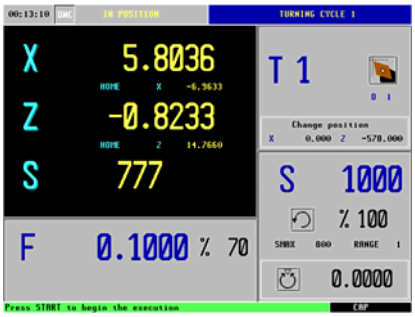
6. Select Z then Cycle Start so this axis is repositioned first.



7. Select X and then Cycle Start to reposition your X Axis.



8. You should now be at your last executed block. Now press Cycle Start to begin executing the rest of your program.



QUICK REFERENCE GUIDE (TC)

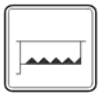
F1: TOOL CALIBRATION

F2: TURNING CYCLE

F3: FACING CYCLE

F4: TAPER CYCLE

F5: ROUNDING CYCLE



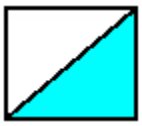
F6: THREADING CYCLE

F7: GROOVING CYCLE

PROFILING CYCLE

DRILLING/TAPPING

POSITIONING CYCLE

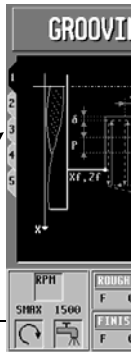


Used to change square symbols.

In Execution mode, by pressing the half-key you will see a different screen that will show your commanded value, actual value, and distance to go value.

Press **SHIFT** then **ESC**.

(Used to switch from conversational to g-code)

Level cycle to select next page in an operation

Change Tabs